

TurnMaster Pro™ 2014

Turning Machine Control and CNC Program Verification For Microsoft Windows™ Based PC's Version 6.0.xxx

The screenshot displays the TurnMaster Pro 2014 software interface. The main window shows a 2D coordinate system with a yellow highlighted area and a magenta tool path. The X-axis ranges from -2.0 to 0.0, and the Z-axis ranges from -0.75 to 0.75. The tool path is a magenta shape that starts at X=0.6, Z=0.1 and moves towards the origin.

The interface includes a menu bar (File, Edit, View, Control, Run, Wizards, Options, Help) and a toolbar with various icons. On the right side, there is a coordinate display showing X: 0.6000 and Z: 0.1000. Below this, the 'Current File Info' section shows the G-Code File as 'SAMPLEPART.CNC' and the Setup File as '1236 MN400 Test.tps'. The 'Outputs' section has checkboxes for Spindle, Coolant, Vacuum, AUX 1, AUX 2, Chuck, and AUX 3, all of which are currently disabled.

At the bottom, there is a G-code editor showing the following code:

```
Line #   Mode   Move   Unit   Spindle   Feed   Loop
-----
61      ABS    G00    IN     0         30     0

/* {X START}: 0.6
/* {Z START}: 0.1
m06 t2 /test 1
m06 t1 /test 2
G90    /SET PROGRAMMING MODE
G74    /SINGLE QUADRANT
```

The status bar at the bottom indicates 'TOOL #1 60 DEG. RH TURNINI | STATUS: Idle' and the date/time '12/23/2013 | 11:47 AM'.

Table of Contents

[Minimum System Requirements](#)

[Installing TurnMaster Pro](#)

[Starting TurnMaster Pro](#)

[Getting Started with TurnMaster Pro](#)

[Menu Reference](#)

[Toolbar and Icon related commands](#)

[Function Detail Reference](#)

[Function Quick Reference](#)

[Configuration and Setup](#)

[Tutorial](#)

[Using the Math Formula Evaluation Feature](#)

[Customizing TurnMaster Pro](#)

[MN400 Controller Info.](#)

[Appendix A - Keyboard Scan Codes](#)

[Appendix B - MetaCommand Reference](#)

[Appendix C - Wiring Diagram for External Cutting Speed Control](#)

[Appendix D - Threading Example using G33](#)

[Appendix E - Using the Configuration File Lathe.TPS](#)

[Appendix F - Using Machine Parameters as Read-Only Variables](#)

Minimum System Requirements



- A Microsoft Windows PC
- For OptiStep or QuickPhase controllers:
Windows 98SE with Internet Explorer 5.5 installed
and an available ISA slot
- For MN400 controller:
Any version of the Windows OS and one free serial or USB port
- 30 megs of free hard drive space
- Minimum 128 megs RAM for Windows SE, 512 for Windows XP,
and 1 Gig for Vista
- Minimum 300MHz Processor for Windows SE, 500MHz for Windows XP,
and 1GHz for Vista

Note that **OptiStep** or **QuickPhase** control cards require a computer with the older **ISA** slot and **Windows 98SE**. For support of newer computers using **USB**, please choose the **MN400** controller or an intelligent **Driverack** system. If you would like to upgrade you legacy systems, **MicroKinetics** offers an upgrade path for your older system. Inquire at www.microkinetics.com

Installing TurnMaster Pro 2014

**** Exit ALL programs and disable ALL Anti-Virus before installing ****

From the Windows Desktop:

- 1) For **Windows XP™**, **Vista™** and future **Windows** releases, log in as an **Administrator**. If you are logged in as a **Standard User**, some files may not copy correctly.
- 2) Insert the **TurnMaster Pro 2014** installation CD. Depending on your PC's *AutoPlay* settings, the following screen may appear.



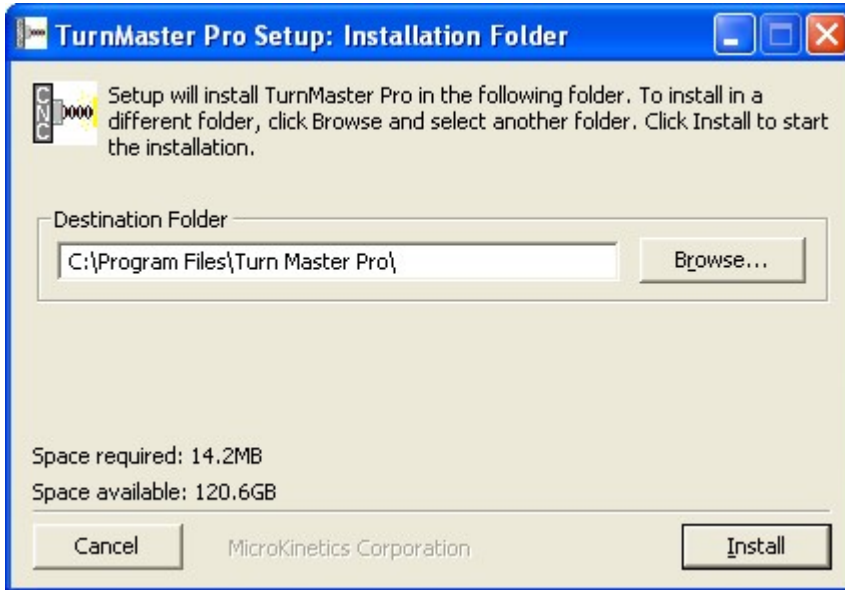
If you see an **AutoPlay** screen with a list of choices, choose "**Run mk_install.exe**" to see this screen.

If nothing happens when you insert the installation CD, open *Windows Explorer* manually and double-click **MK_Install.exe** to see the this screen. Alternatively, you can double-click **Tmpfw_Install.exe** and continue to **Step 4**.

Note: If you are installing the latest update from the **MicroKinetics** website, navigate to <http://www.microkinetics.com/tmpfw/instruct.htm> and follow the instructions.

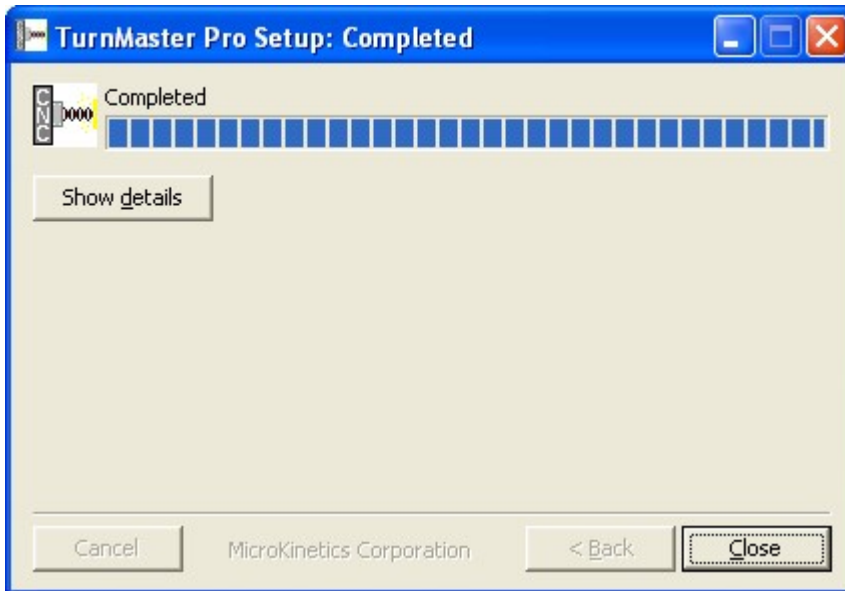
- 3) You may view documentation or install the software. If you choose **Full Install** or **Update**, continue on to **Step 4**. Otherwise press the **Exit** button to close the install program.

4) The following screen (or a similar screen if **Update** was selected) will appear:



You may accept the default Destination Folder, or select a different one. Click the **Install** button. This starts the Auto-Install process.

5) After the installation process is completed, the following screen will appear:



TurnMaster Pro is now installed on your hard drive.

**** NOTE:** If you experience any difficulties or errors when installing, write down detailed information regarding the problem, including any cryptic error information, before calling Tech Support.**

**** This will enable us to solve your problem as efficiently as possible. ****

Starting TurnMaster Pro 2014

IMPORTANT: Due to precise motor control timing you must disable/close **ALL** Instant Messaging and Virus protection software before running **TurnMaster Pro**.

After Installation is complete, you must re-start your computer. Then you can run the program by selecting '**START**' ; '**PROGRAMS**' and then clicking on the **TurnMaster Pro** icon in the program list.

TIP: If you right click the icon in the program list, a menu will appear with an option to "**Send To -->**". Put your mouse pointer over this item and another menu will pop out with an option that says "**Desktop as Shortcut**". Put your mouse over this option and left click.

Windows has now placed a **TurnMaster Pro** Icon onto your desktop for easy access.

When you first start **TurnMaster Pro** you will see the following screen:



You will notice the evaluation message displayed on this screen until you 'Unlock' the software. This can be done by using the File Menu Bar at the top of the **TurnMaster Pro** Main Screen; clicking the word **Help**, this will drop down a small sub-menu; click the option that says **Unlock Software**.

A small window will appear and contain a **System ID #** and have a blank space to enter an **Unlock Code**. To obtain an Unlock Code, call our sales department @ (770) 422-7845 or email inquiries to sales@microkinetics.com.

Once you receive your Unlock Code, you enter it into the space provided and click the button labeled **Unlock Software**.

Now when you start **TurnMaster Pro 2014** the initial screen will look like this:



After the initial splash screen shown above, if you have Norton Antivirus or McAfee VirusShield installed on your system, you may see the following screen:



This screen will appear each time you start **TurnMaster Pro**, unless you check the option at the bottom of the screen that says 'My Anti-Virus is disabled, skip this message in the future'. This screen is just a reminder, so if you do check the option to skip the message in the future and do not disable your Antivirus software, you will not be able to perform machining functions.

TurnMaster Pro only detects Norton Anti-Virus or McAfee VirusShield. If you are running another brand of Anti-Virus, this message will not be displayed. You will need to disable ANY virus protection, regardless of manufacturer before machining.

Also note that for Windows versions later than Windows 98, **TurnMaster** will not check for anti-virus software, and thus this screen will not be displayed.

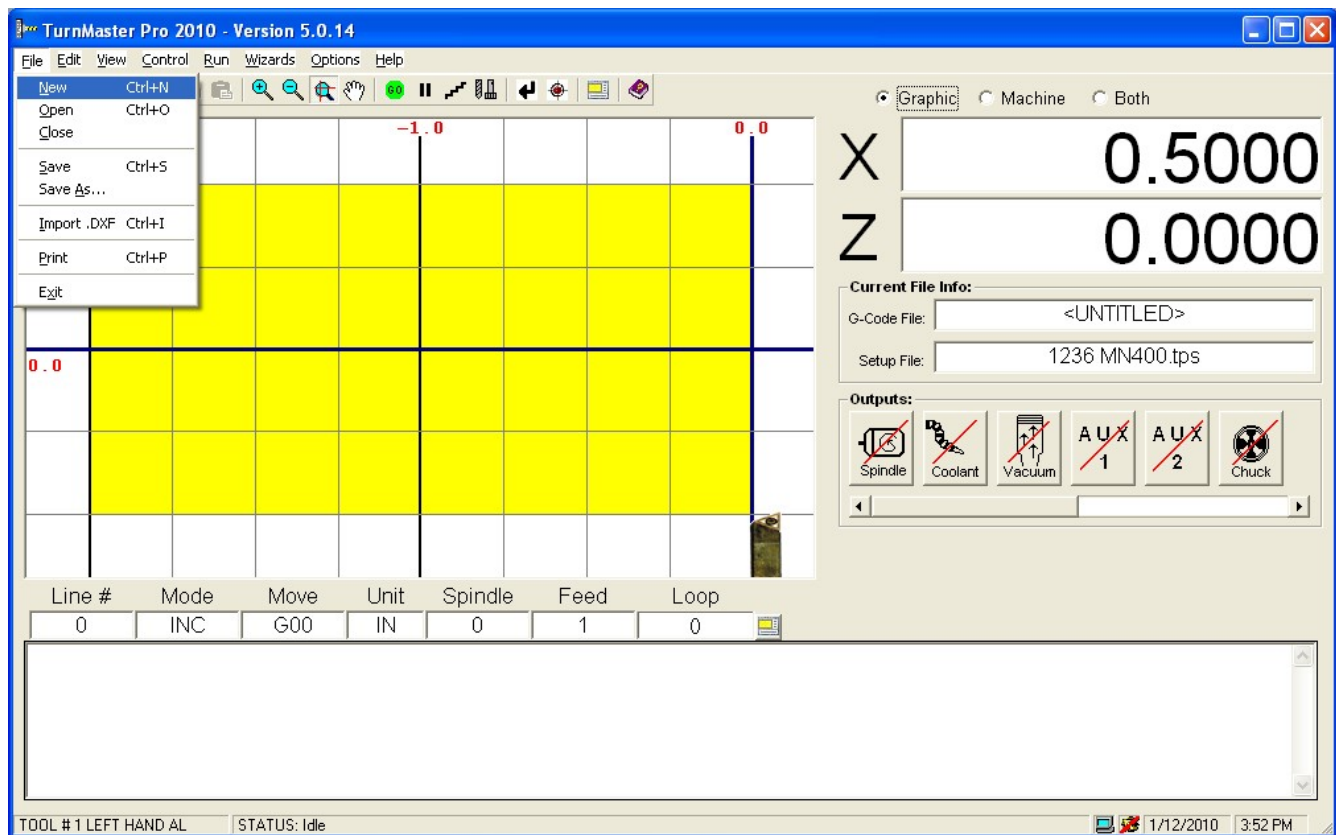
Norton Antivirus and **McAfee VirusShield** are trademarks of their respective companies.

Getting Started with TurnMaster Pro 2014

Thank You for purchasing **TurnMaster Pro 2014 G-code Interpreting Software**. This is an installation and command reference manual for **TurnMaster Pro** that will help you with all aspects of the program. Observe the top line of the **TurnMaster Window**. Each word is a heading for a pull-down menu.

The pull-down menu will appear when the heading is selected by the mouse or keyboard. To access a pull-down menu, use the mouse to position the arrow on top of a pull-down menu heading and click the left mouse button. Alternatively, you can use the keyboard to access a pull-down menu by pressing **<ALT>** and the first letter of a heading.

Once the menu is in view, you can select any of its options by positioning the arrow on the option and clicking the left mouse button.



Some of the selections have **Hot Keys** associated with them. Hot keys allow quick access to a command through the keyboard and they are indicated to the right of the selection. For example, the hot key used to open a part program is **<CTRL>-O** which means holding down the **<Ctrl> key** and pressing **O** simultaneously.

The underlined letters are called **QuickSelect** keys. These allow access to a command only while the pull-down menu is displayed. Some of the selections in the menu have corresponding icons on the toolbar below the menu heading. These allow the user to rapidly execute a command without using the pull-down menus. Clicking these icons executes the same command as using the pull-down menus.

In the **Machine Parameters Window**, the menu is setup with **Tabs**, just like a filing cabinet. When the mouse moves over these tabs, they become highlighted. This is called **Mouse-over Hot Keys**. Click on a tab to show its relevant information.

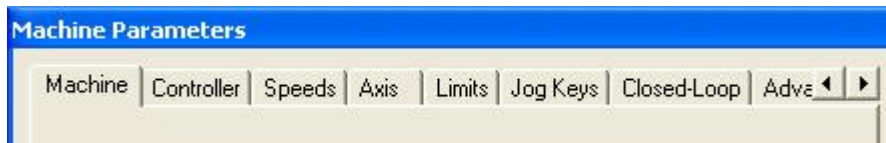
The following graphics show the Machine Parameter Tabs. The graphic on top demonstrates what the tab looks like when the **Mouse-over Hot Key** is **active**.

The graphic on the bottom demonstrates what the tab looks like when the **Mouse-over Hot Key** is **inactive**.

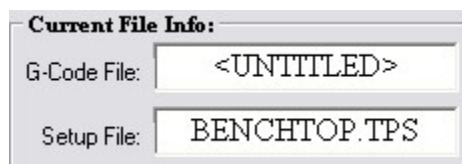
Active:



Inactive:



Some menus can be accessed rapidly with a **Mouse QuickSelect** click. These allow access to the **Open File Dialog Window** and the **Machine Parameters menu**, by clicking in the white space of the G-code File or the Setup File boxes in the Current File Info frame. **(Shown below)**



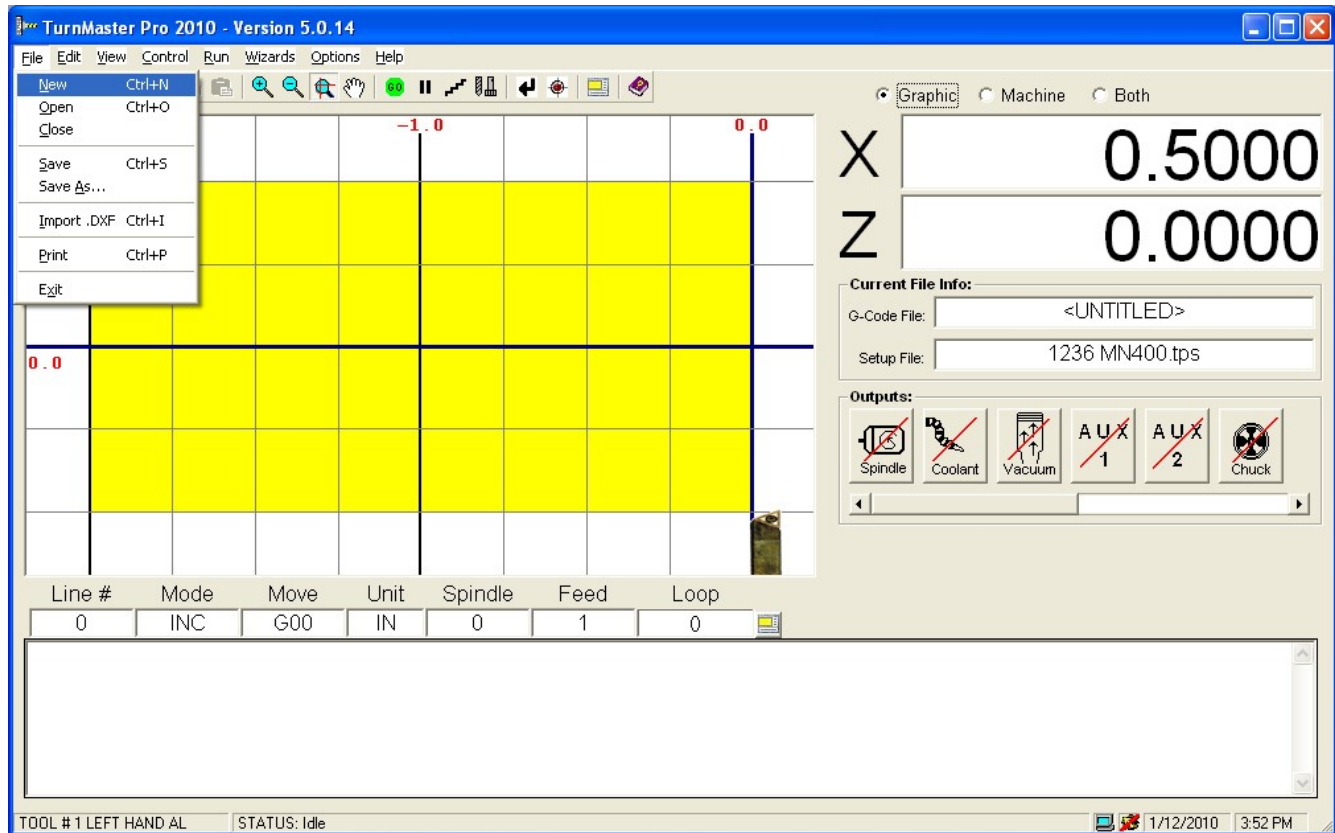
The **Zoom-In**, **Zoom-Out**, and **Pan** functions are also accessed by left clicking and dragging the mouse over an area in the cutting window.

If the mouse looks like a crosshair when over the XY cutting window, you are in **Zoom** mode. You may drag the mouse over a an area, and **TurnMaster** will re-zoom the XY cutting window to most closely match the area specified.

If the mouse looks like a hand when over the XY cutting window, you are in **Pan** mode. Clicking and dragging the mouse will pan the XY cutting window in the direction and distance specified.

Menu Reference

The graphic below represents the Menu Bar with the "File" pull down menu accessed.



File- New

Clears the part program in memory and establishes the standard defaults for a new part program. The first icon in the Toolbar also corresponds to this menu command.

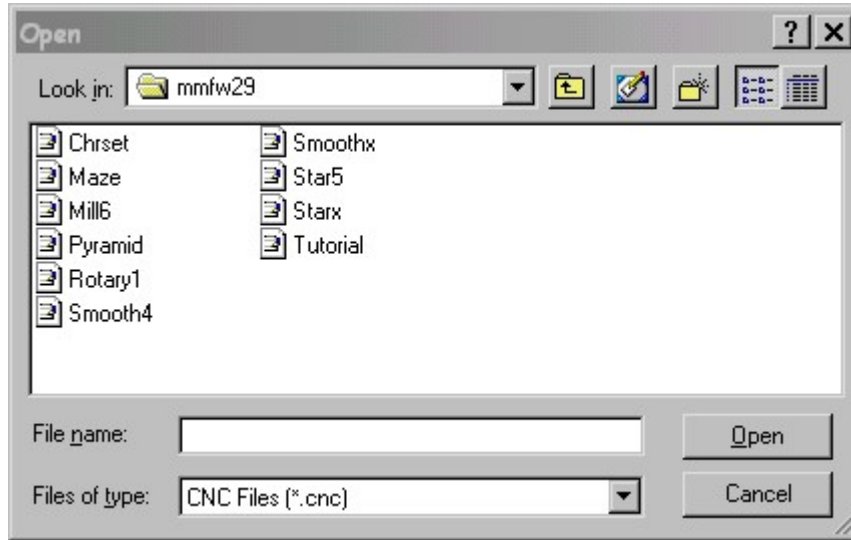
File- Open

Displays the Open File Dialog Window which displays all part programs in the current directory. The second icon in the Toolbar also corresponds to this menu command. The graphic below represents the Open File Dialog Window.

You can easily load your favorite part program by Dragging and Dropping the .CNC file onto the **TurnMaster Pro** icon that is used to start the program. **TurnMaster Pro** must not be running when this feature is used. After dragging and dropping the file onto the **TurnMaster Pro** start icon, **TurnMaster Pro** will startup normally and the file that was dropped onto the icon will be automatically loaded.

TIP:

Just click in the whitespace of the G-code file box in the Current File Info. frame on the Main Screen for rapid access to the Open File Dialog Window



File- Close

Closes the current .CNC file in memory, but does not restore the counters or the cutting screen.

File- Save

Quickly saves the current part program to disk. The extension CNC is automatically assigned. The third icon in the Toolbar also corresponds to this menu command.

File- Save As...

Displays the Save File Dialog Window, which Saves the current part program to disk allowing the user to rename the file. The file extension .CNC is automatically assigned.

File- Import .DXF

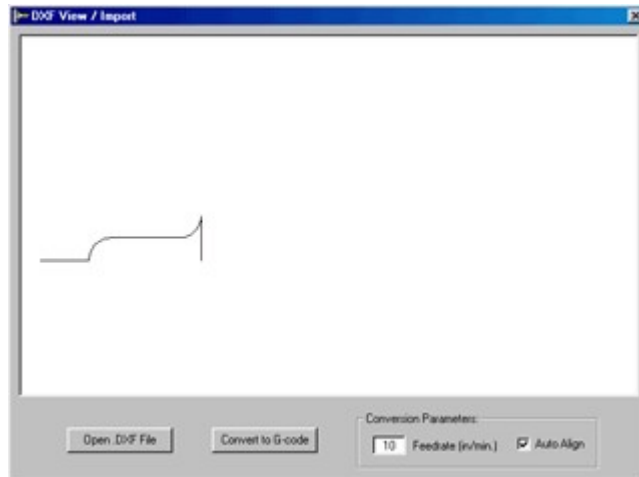
This menu command allows you to import **industry standard .DXF files up to version 15 (AutoCAD 2000 format)**. This is the latest **AutoDesk** release.

The following Entities only, are supported for conversion to g-code:

- POLYLINE
- LINE
- ARC

****NOTE:** AutoCAD does true circular interpolation whereas Corel Draw breaks circles and arcs into coordinates and is therefore not as smooth and continuous when machining. Also, this function does not accept .DXF files that are in 3D format or that have tables enabled.

When selected, an **Open File** dialog box will appear. Select the .DXF file of your choice and click 'Open'. A new window will appear with a large viewport (image below). This is the **DXF viewing conversion window**. At the bottom of the window you will notice two command buttons, and a group of conversion parameters. Once the file is opened, it will appear in the viewport. Input your conversion parameters into the areas provided and select "**convert to G-code.**" A **Save File** dialog box will appear next. Select a drive, directory and name to save your file under and click 'Save'. You have just created a .CNC file for use with **TurnMaster Pro**.



****NOTE:** TurnMaster will automatically name your G-code file with the same name as your .DXF file unless you specify otherwise.

TIP: Remember to test your program Graphically before machining to prevent potential damage to your machine.

Conversion Parameter details:

Feedrate:

This parameter specifies the feedrate for the **conversion utility**. This is just an **initial value**, and may be changed with the **CNC Editor**.

Auto-align:

This parameter specifies the auto alignment feature for cutting. If the values contained within your **.DXF file** are negative, the cut would be taken off the viewable screen. When checked, the conversion utility will add a **G92** specifying the positive X and Y quadrant (upper right), for all moves.

Auto-Adjust Material Size:

This parameter specifies whether TurnMaster should automatically change the current material size needed to cut the current imported **.DXF** file. If selected, when you return to the **main screen**, the material size will be adjusted automatically.

Auto-Load Program:

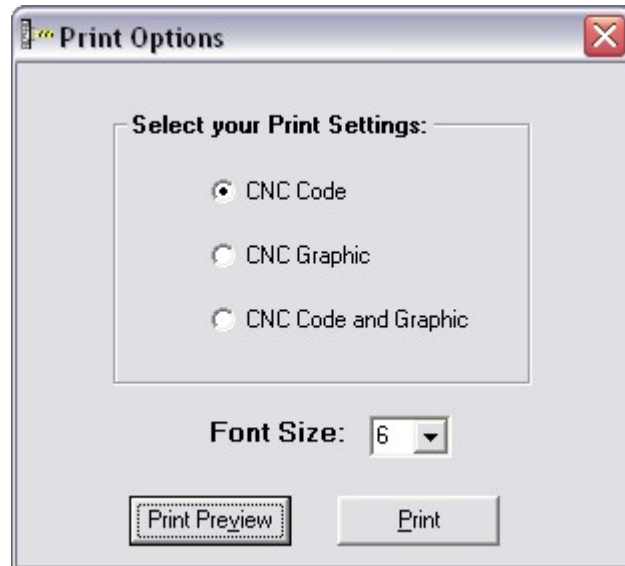
This parameter specifies whether TurnMaster should automatically load the program into memory. If selected, when you return to the **main screen**, the program will be loaded.

File- Print

Displays the Print Dialog Window, which allows the user to choose Print Options, and utilize the built-in Print Preview Function.

The fourth icon in the Toolbar also corresponds to this menu command.

The graphic below represents the Print Dialog box.



File- Exit

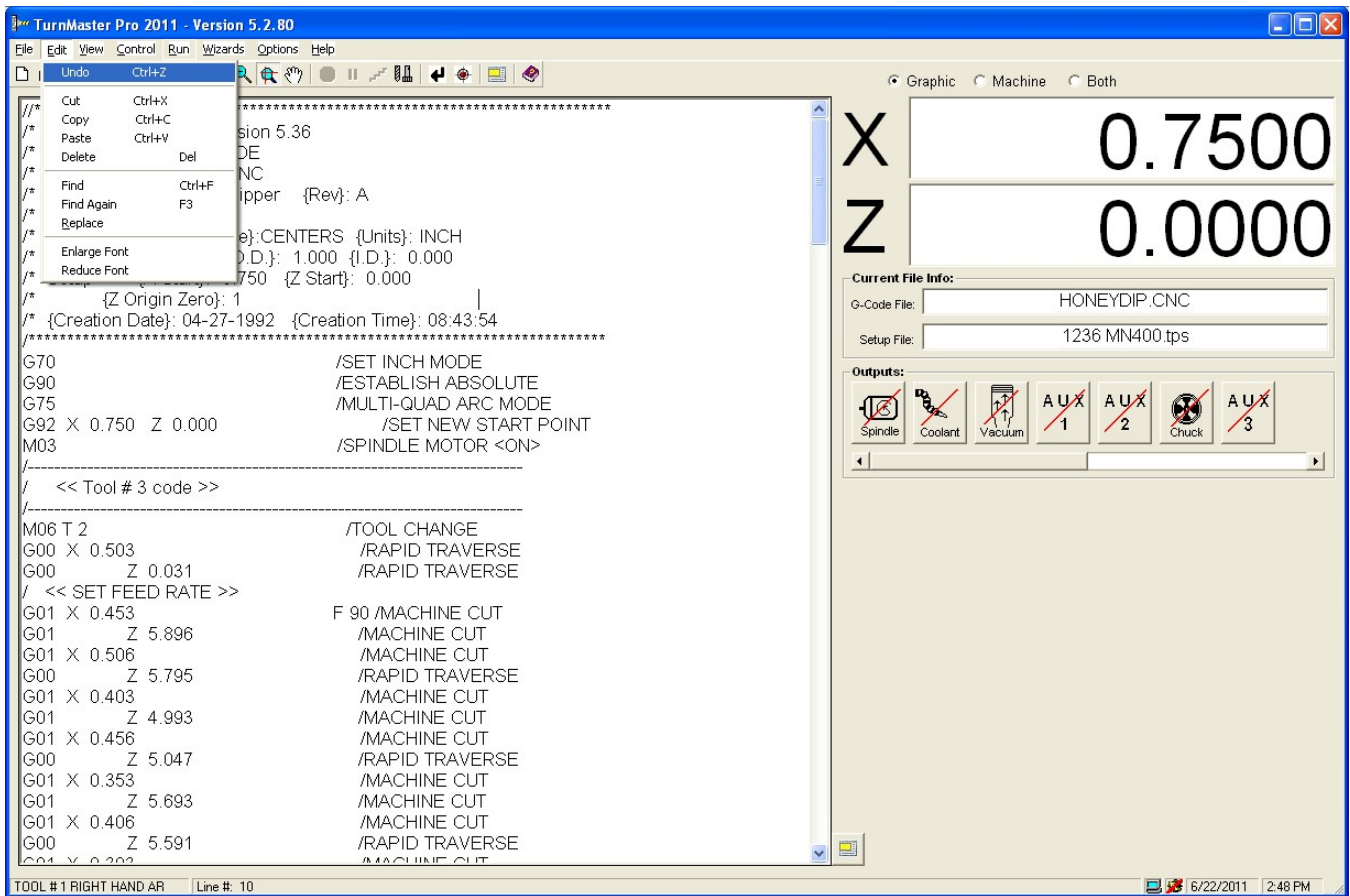
Exits **TurnMaster Pro** and returns to the Windows desktop.

Selecting the **Edit Menu** produces the list of Editing selections. The commands on this menu are Standard Text Editing commands seen in today's popular text editors. Some of the selections have **Hot Keys** associated with them. **Hot Keys** allow access to a command quickly through the keyboard and they are indicated to the right of the selection.

The underlined letters are called **QuickSelect** keys. These allow access to a command only while the pull-down menu is displayed.

**** NOTE:** The Edit Menu is only enabled when in full-screen **CNC Edit Mode**. See the View Menu for more information on **CNC Edit Mode**.

All the Edit Menu commands are grayed out and disabled unless **TurnMaster Pro** is in CNC Edit Mode (Full-Screen Editor). ******



Edit- Undo

Undoes the last edit (insert, delete, or change). Only the last edit can be undone.

Edit- Cut

Cuts the currently highlighted text out of the page and onto the clipboard. The fifth icon in the Toolbar also corresponds to this menu command.

Edit- Copy

Copies the currently highlighted text onto the clipboard. The sixth icon in the Toolbar also corresponds to this menu command.

Edit- Paste

Pastes the current data from the clipboard to the current cursor location. The seventh icon in the Toolbar also corresponds to this menu command.

Edit- Delete

Deletes the currently highlighted text permanently.

Edit- Find

Finds a specified string of characters in the CNC Edit window.

Edit- Find Again

Finds the last sequence of characters specified using the FIND command, in the CNC Edit window. This command may only be used AFTER using the FIND command. Once the entire CNC Edit window has been searched, you will receive the message 'Word not found again'. To utilize this command again, you must specify a sequence of characters with the FIND command.

Edit- Replace

Finds a specified string of characters in the CNC Edit window and replaces that string with a different string. The user will have the option to replace only the first occurrence of the original string or all occurrences.

Edit- Enlarge Font

Enlarges the font on the CNC Edit window. Each time this option is selected the font size is increased by 1.5 pt, up to a maximum size of 16 pt.

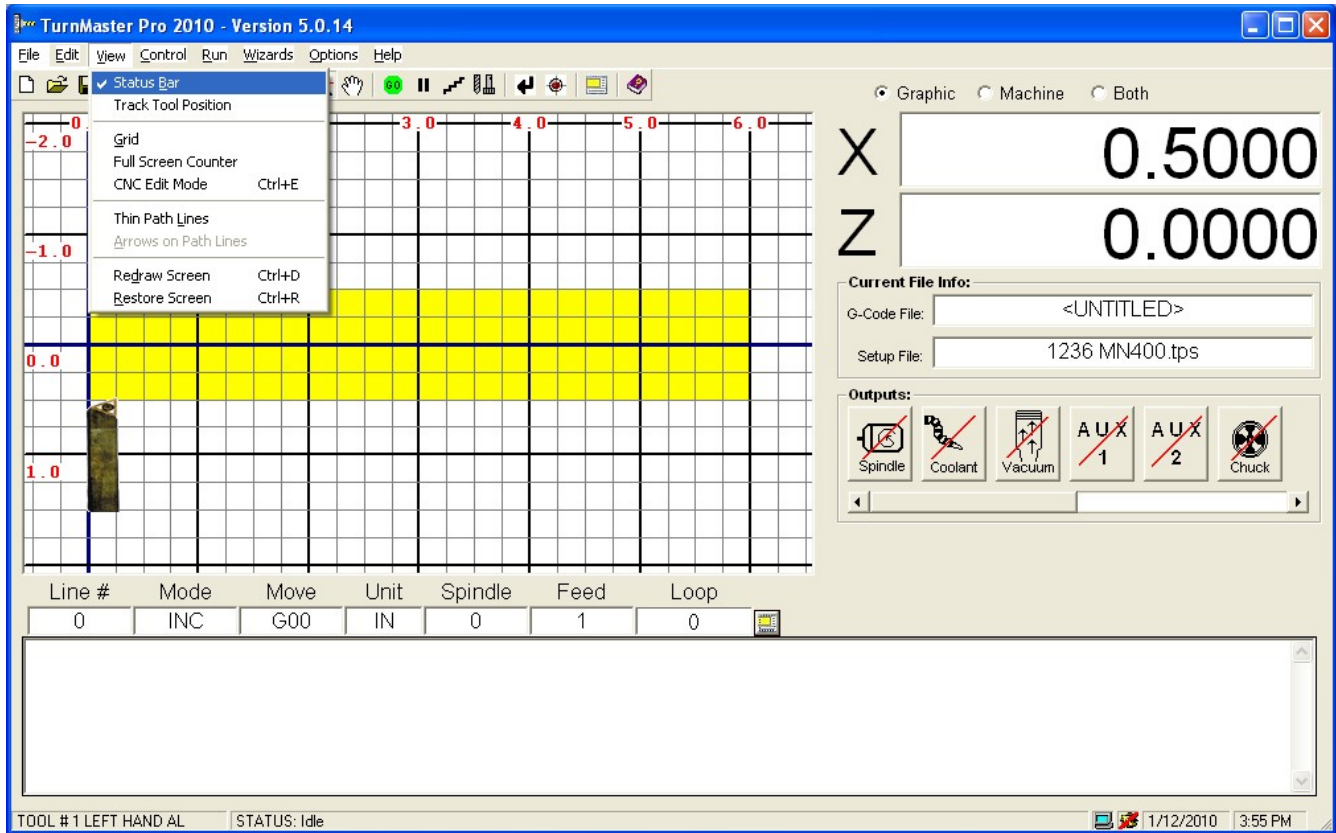
Edit- Reduce Font

Reduces the font on the CNC Edit window. Each time this option is selected the font size is decreased by 1.5 pt, down to a minimum size of 8 pt.

Selecting the **View Menu** produces the list of visual selections. Some of the selections have **Hot Keys** associated with them. Hot keys allow access to a command quickly through the keyboard and they are indicated to the right of the selection.

For example, the Hot key used to restore the screen is **<CTRL>-R**. This is done by holding down the **<CTRL>** key and pressing **R** simultaneously.

The underlined letters are called **QuickSelect** keys. These allow access to a command only while the pull-down menu is displayed.



View- Status Bar

Allows the user to either enable or disable displaying a status bar at the bottom of the window with current tool selection, current program status, current program mode, and current system date/time. When a check mark appears next to this function, it is enabled.

The graphic below represents the status bar.



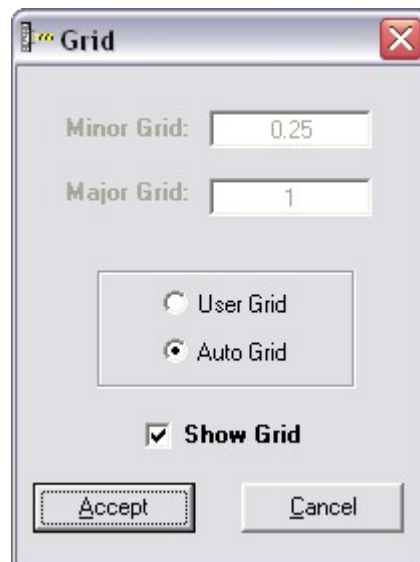
View- Origin Marker

Allows the user to either enable or disable displaying the blue, crosshair Origin Marker over the tool in the XY viewport. When a check mark appears next to this function, it is enabled.

View- Grid

Displays the **Grid Window** which allows the user to select the settings for the Minor and Major grid lines, use the **Auto-Grid function**, or turn **On/Off** the Grid. When a check mark appears next to this function, it is enabled.

The graphic below represents the **Grid menu**.



View- Full Screen Count

Displays the **XZ** Counters in **Full Screen Format**, while temporarily hiding all other controls. When a check mark appears next to this function, it is enabled. The graphic below represents the **Main screen in CNC Edit mode (left)** and the **Full Screen Count mode (right)**

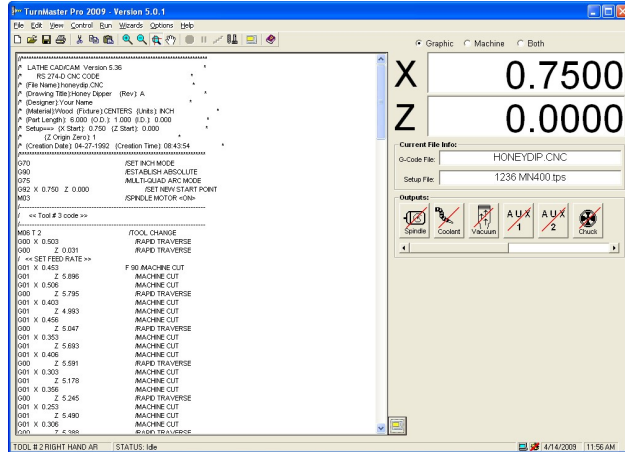
View- CNC Edit Mode

Toggles CNC text edit mode. When on, **TurnMaster** displays the CNC text editor while temporarily hiding the cutting window. This allows the user to Edit any currently loaded program, or enter a part program manually. This works like any standard Text editor. When a check mark appears next to this function, it is enabled.

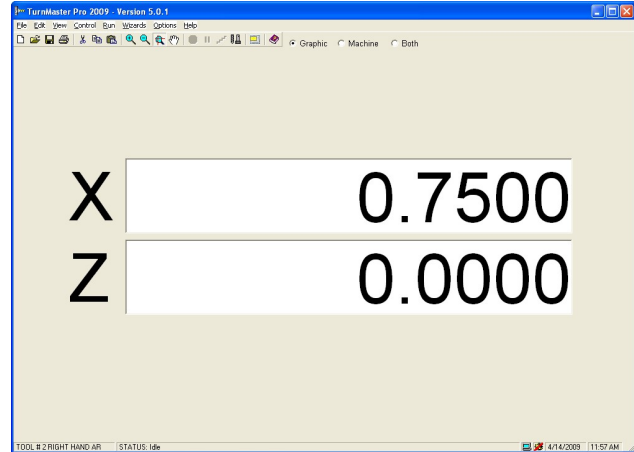
Note that the **CNC Edit Mode** can also be toggled via the **<Ctrl> E** hotkey, or the small image of the application on the right side of the CNC textbox.

The graphic below represents the Main screen in **CNC Edit mode (left)** and the **Full Screen Count mode (right)**.

Main Screen with CNC Edit mode enabled



Main Screen with Full Screen Counter mode enabled



(These graphics have been reduced in size for spacing purposes)

View- Thin Path Lines

Normally, the Graphics screen shows the Tool Path using the actual thickness of the current tool. Sometimes, however, you may want to see the tool path without needing to see a realistic view of the resulting part. You can check this menu item for this function.

View- Arrows on Path Lines

This checkbox is only available if **View | Thin Path Lines** is checked. If checked, a small arrow will be drawn at the end of each path line to show the direction of the path.

View- Redraw Screen

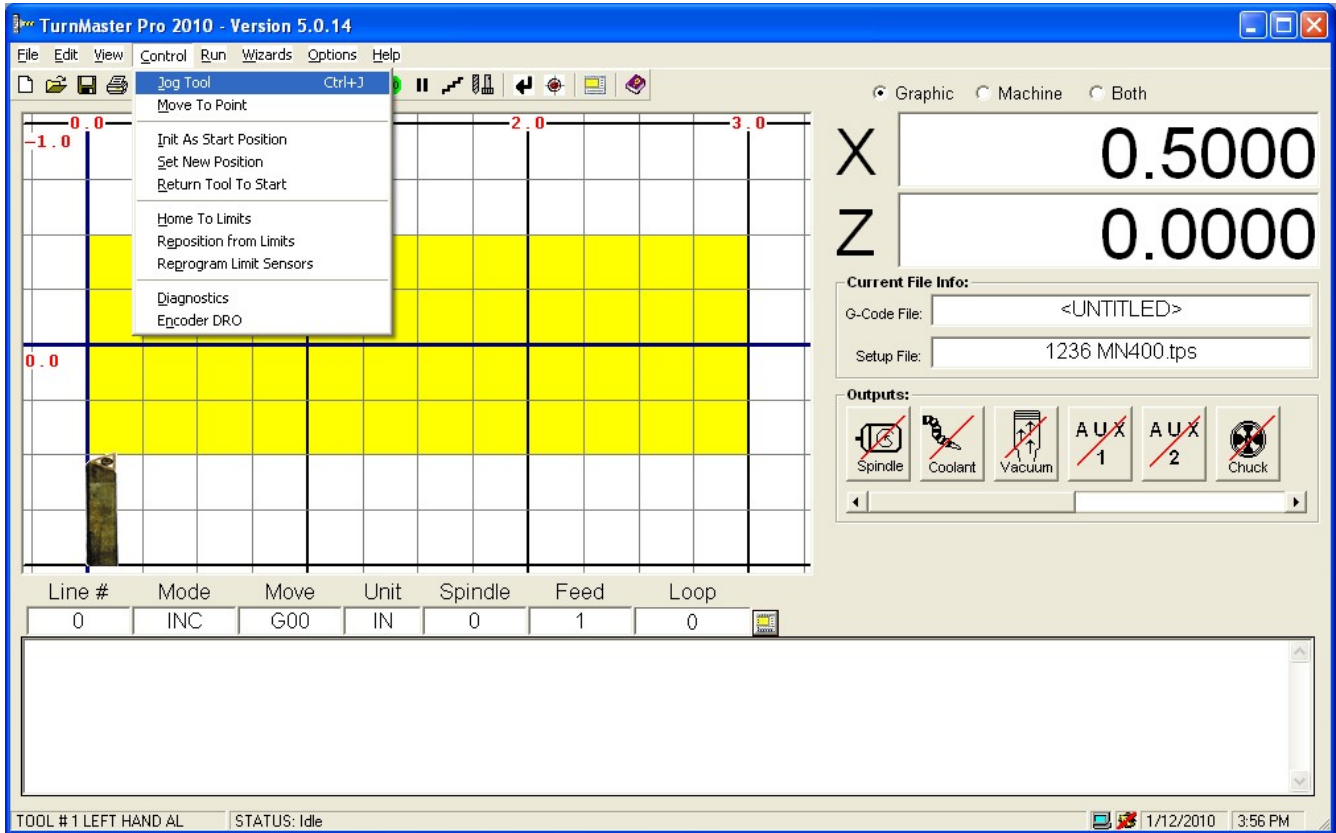
Redraws the screen. Any previous graphical run will be cleared, but the size and location of the viewport will be retained.

View- Restore Screen

Redraws the screen to show the entire uncut raw stock. Choosing this option when in 'Graphics' mode resets the graphical XZ counters. When in 'Machine' mode, only the machine counters are reset and when in 'Both' mode, both the graphical and machine counters are reset.

The values in the Material Size menu are applied here.

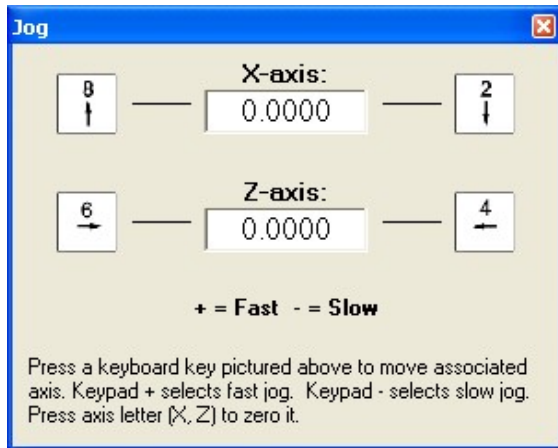
The **Control Menu** is used to jog the tool of the machine, set up the starting position of the machine, and calibrate the machine. The underlined letters are called **QuickSelect** keys. These allow access to a command only while the pull-down menu is displayed.



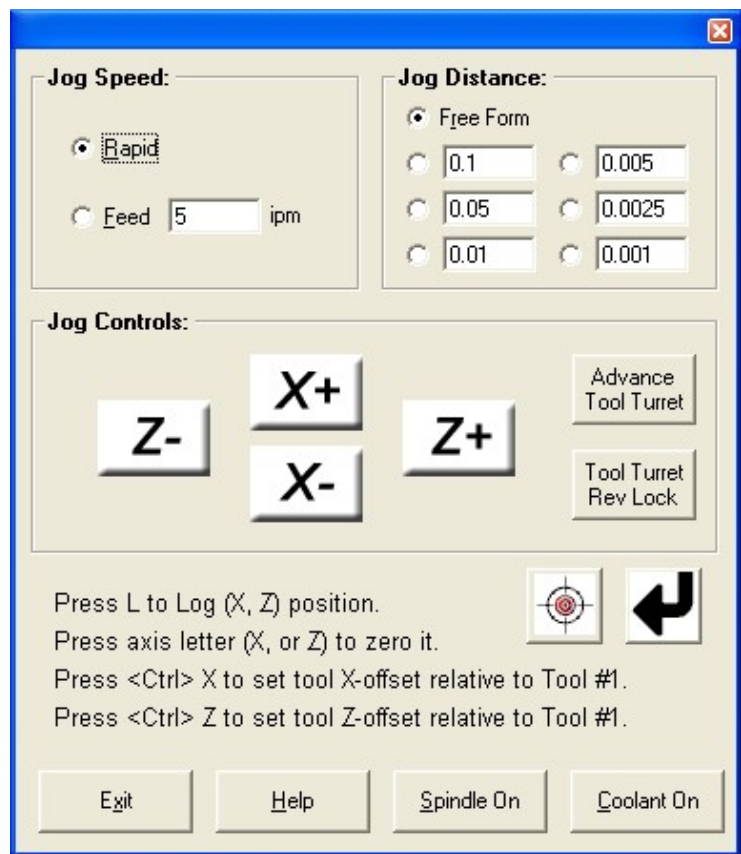
Control- Jog Tool

Allows the user to jog all axes on the machine. There are two different jog screens, depending on the controller you selected. Instructions for each screen are displayed.

A machine must be correctly connected to use this function. For both jog screens, pressing the corresponding axis letter (X or Z) will zero that axis at its current location.



OptiStep Plus & QuickPhase Controllers

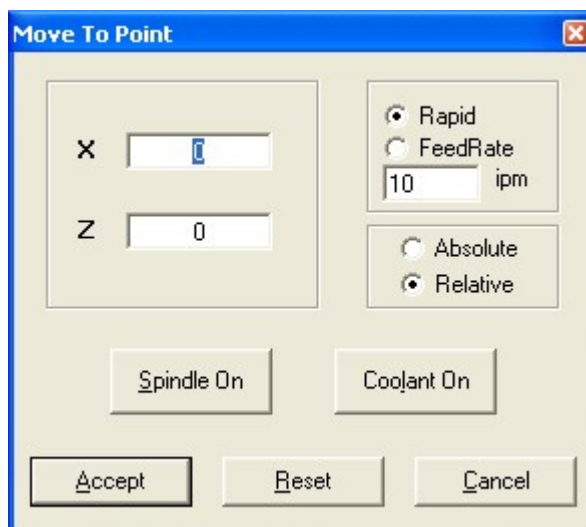


MN400 Controller

Control- Move To Point

Positions the currently selected tool to the **XZ** coordinates entered in the Move To Point Menu. The program will reflect a cutting path **ONLY** if the **X**-axis is within the surface of the part (**a negative number.**)

You can select either rapid moves or specify a feedrate in inches per minute. You can also choose between absolute and relative moves.



Control- *Init. As Start Position*

Initiates the current position of the tool as the Start Position and resets the internal counters. The current location of the tool is assumed to be in the tool start position.

Control- *Set New Position*

Sets the location entered by the user and sets the internal counters to the new values. This does **NOT** affect the options Tool Start Position setting.

Control- *Return Tool to Start*

Moves the tool to the Tool Start Position. If **Machine Mode** is selected, the tool will move to the tool start position. This does **NOT** reset the internal counters.

Control- *Home to Limits*

Moves the machine to the limit sensors indicated in the Options-Limit Sensors Dialog Box. Limit switches must be connected to use this function.

Control- *Reposition from Limits*

Returns tool to its Home Limit switches, clears counters and then positions tool at the start point. You must use Reprogram from Limit Switch command to set the distances from each limit switch to the start point before using this command. Limit switches must be connected to use this function.

This is also a good way to see if your machine is running within tolerances. An expected value of .002" or less should be assumed for tolerances.

Control- *Reprogram Limit Sensors*

Automatically measures and stores the distance from the Start Point to each of the 2 Limit switches. The tool must be positioned at its Start Point prior to selecting this function. Limit switches must be connected to use this function.

Control- Diagnostics

This allows the user to find the maximum jog and rapid traverse speeds of any or all axes on the lathe. It is a self-running test that exercises the selected axes to determine the speed limits of the machine. The maximum speeds that are found by this function can be automatically entered into the **Position and Feed Speeds** window.

A file named DIAG.LOG that contains test information is created in the current directory and is a standard text file that can be read using any text editor. This option uses the home limit switches selected in the **Limit Sensors** option (if **<NONE>** is selected for any axis, this test can not be performed for that axis).

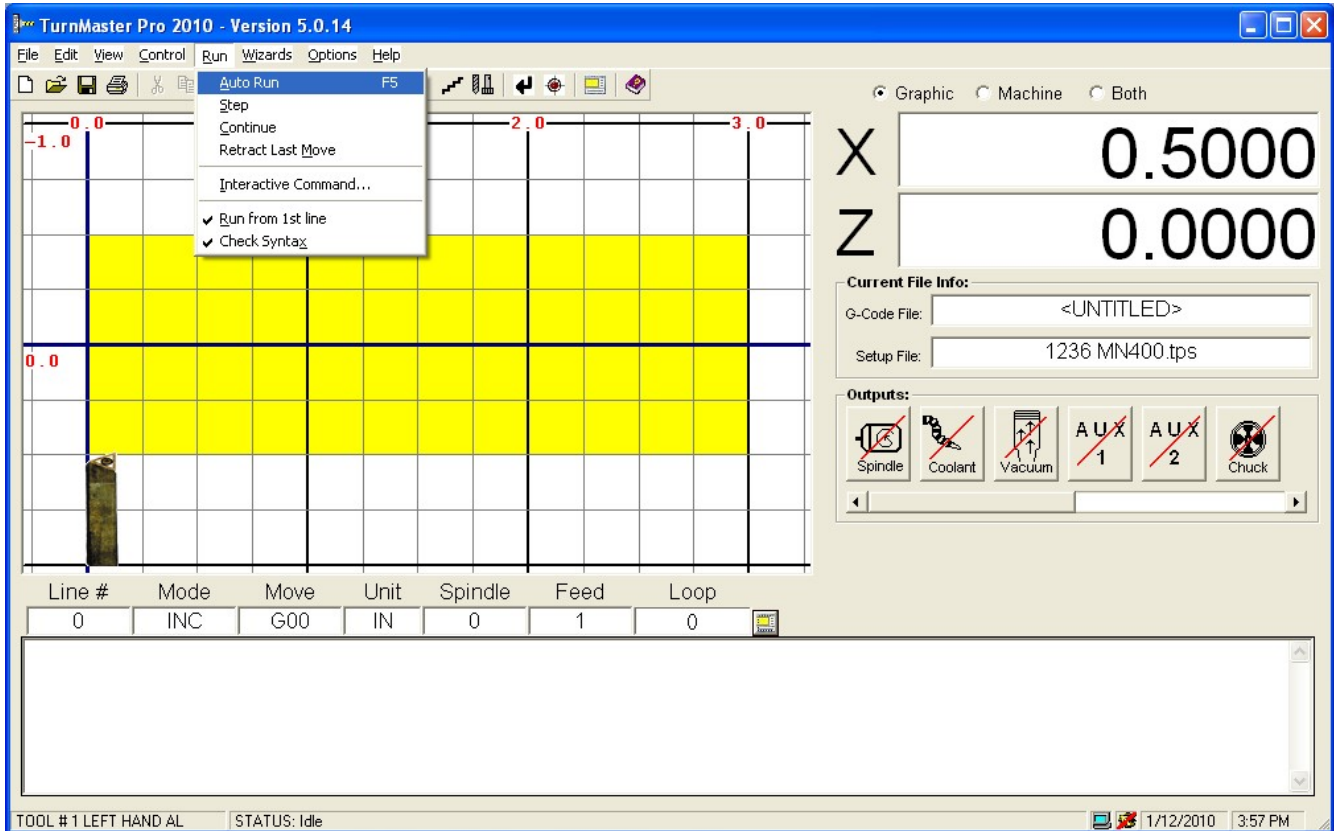
The screenshot shows a software window titled "Diagnostics" with a blue title bar. Inside the window, there is a central instruction box with the following text: "Please set Feed Rate to 100% before running Diagnostics. Do not adjust Feed Rate during these tests. Press any key to pause or abort the test while running." Below this, the "Axes to Test" section contains a table with checkboxes and input fields for "Unramped Distance" and "Ramped Distance". The X-Axis is selected with a checked box, showing 0.5 for Unramped Distance and 4 for Ramped Distance. The Z-Axis is not selected with an unchecked box, showing 0.5 for Unramped Distance and 2 for Ramped Distance. To the right of this table are four input fields: "Start Speed (steps/sec):" with a value of 1000, "Speed Change (steps/sec):" with a value of 200, "Acceleration (steps/sec^2):" with a value of 5000, and "Tolerance (inches):" with a value of 0.005. At the bottom left is a "Diagnostics Message Window" with a scroll bar. On the bottom right are four buttons: "Begin", "Show Last", "Print", and "Exit".

Axis	Unramped Distance	Ramped Distance
<input checked="" type="checkbox"/> X-Axis	0.5	4
<input type="checkbox"/> Z-Axis	0.5	2

Start Speed (steps/sec): 1000
Speed Change (steps/sec): 200
Acceleration (steps/sec²): 5000
Tolerance (inches): 0.005

Buttons: Begin, Show Last, Print, Exit

The **Run Menu** is used to execute the part program. The part program can be tested graphically or it can be used to actually machine a part. The underlined letters are called **QuickSelect** keys. These allow access to a command only while the pull-down menu is displayed.



Run- Auto Run

Executes the current part program in memory in continuous mode. Either **Graphics Mode** or **Machine Mode** must be enabled to use this option. The twelfth icon in the Toolbar also corresponds to this menu command

Run- Step Mode

Executes the current part program one line at a time pausing after each line. The user must click **<CONTINUE>** to execute the next program line.

Either **Graphics Mode** or **Machine Mode** must be enabled to use this option. The fourteenth icon in the Toolbar also corresponds to this menu command.

Run- Continue

This is used to continue executing the part program at the current instruction. Either **Graphics Mode** or **Machine Mode** must be enabled to use this option.

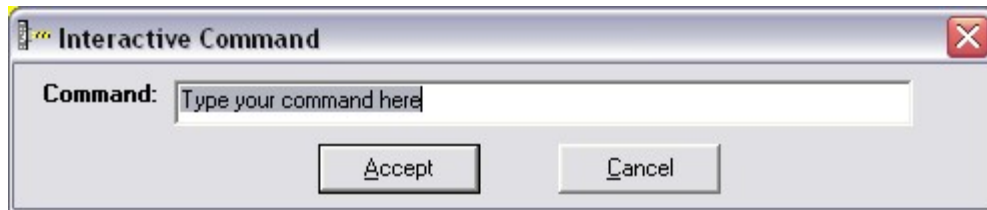
Run- Retract Last Move

This is used to "Undo" the last move executed by the software. This function can only be used in machine mode.

Run- Interactive Command

This allows the user to enter a single program line and see the results immediately following on the screen. This option can **NOT** be used in **Machine Mode**.

The graphic below represents the **Interactive Command Window**.



Run- Run from 1st line

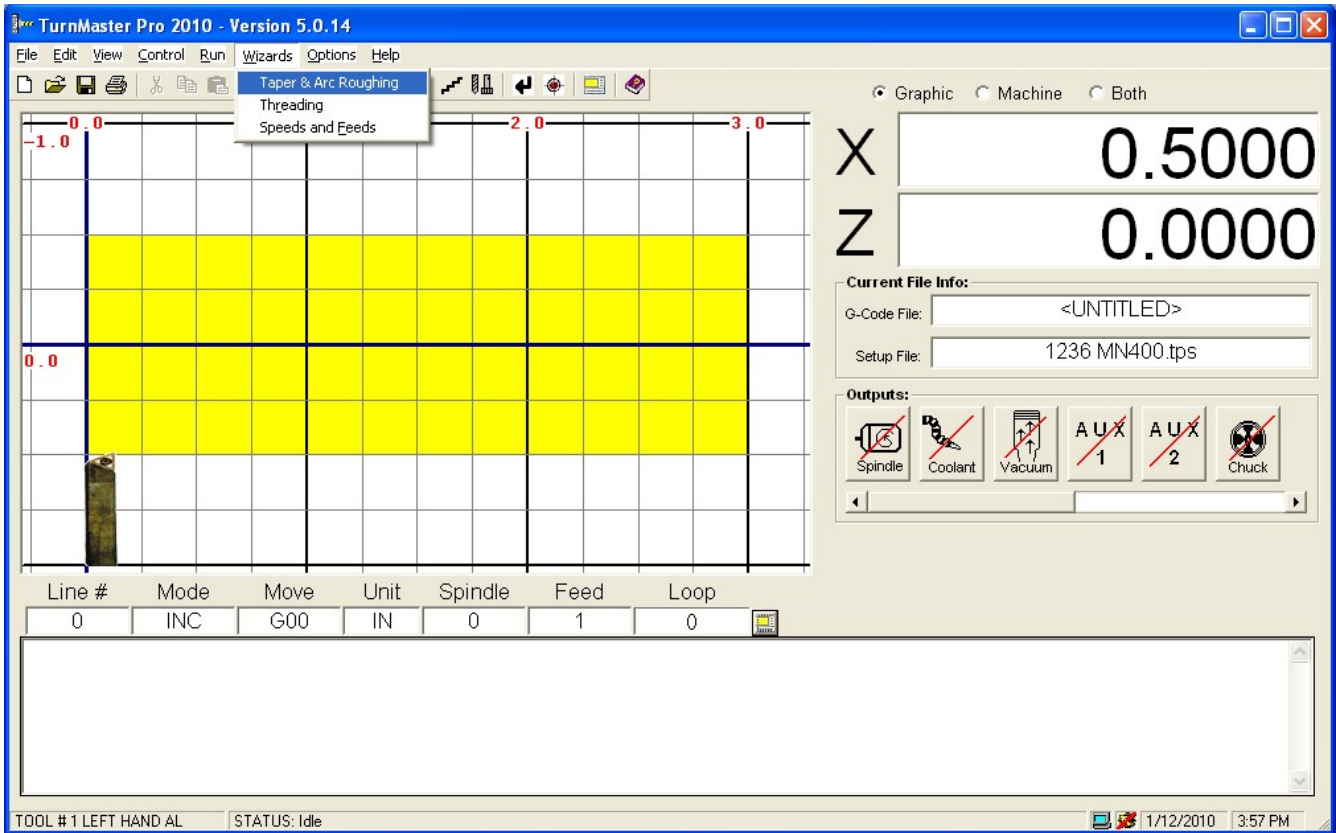
This option allows the user to start the part program at any good line number. When checked, the part program will start from the first line in the file. If unchecked, the software will prompt the user to enter the line number they wish to start the part program from.

Run- Check Syntax

This option allows the user to run their program with automatic syntax checking of their program on. While running the program, if an error is encountered, a dialog box with some helpful hints on how to correct the formatting will pop up.

The **Wizards Menu** gives the user a set of wizards to help in the creation of a part.

The underlined letters are called **QuickSelect** keys. These allow access to a command only while the pull-down menu is displayed.



Wizards- *Tapering & Arc Roughing*

Opens the Taper Window which allows the user to describe a start and end point along the Z axis describing a taper. You can also enter the beginning and ending diameters of the taper as well as the cut depth per pass and the finishing pass clearance. By pressing the Compute button, this routine will calculate the G-code for you and then allow you to copy and paste the code into your CNC editor in the program.

Taper & Arc Roughing

Type of Roughing
 Taper Arc

Type of Arc
 Concave Convex

External Taper Internal Taper

Start of Taper (Z-Axis)
End of Taper (Z-Axis)
 Extend Cuts to: (Z-Axis)

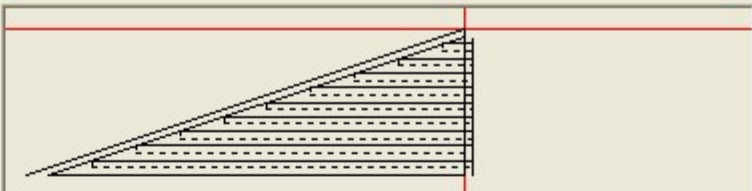
Diameters of Taper or Starting X and Angle
 Diameters Starting/Ending X

Start Diameter
End Diameter
Start of Taper (X-Axis)
End of Taper (X-Axis)
Radius of Arc

Computed Values
Angle of Taper (0-90)
Length of Taper

Cut Depth Per Pass
Finishing Pass Clearance

Bidirectional Cutting ? Display Arrows ?



```
/ External 18.44 degree taper with Unidirectional Roughing.  
G90 /Absolute Mode  
G75 /Multiple Quadrant Mode  
G0 X1 Z0.05 / Move to initial cutting position  
G1 X0.9  
G1 Z-2.55  
G0 X0.95  
G0 Z0.05  
G1 X0.8  
G1 Z-2.25  
G0 X0.85  
G0 Z0.05  
G1 X0.7  
G1 Z-1.95  
G0 X0.75  
G0 Z0.05  
G1 X0.6  
G1 Z-1.65  
G0 X0.65  
G0 Z0.05  
G1 X0.5  
G1 Z-1.35  
G0 X0.55  
G0 Z0.05  
G1 X0.4  
G1 Z-1.05  
G0 X0.45  
G0 Z0.05  
G1 X0.3  
G1 Z-0.75  
G0 X0.35  
G0 Z0.05  
G1 X0.2
```

Wizards- Threading

Opens the Threading Window which allows the user to describe a specific thread. There are many options that will allow the user to specify exactly what type of thread they want to cut. By pressing the Compute button after entering all of the data, the routine will calculate the proper G-code for you which can be cut and pasted into your program with the touch of a button.

The Threading Wizard dialog box is shown with the following settings:

- Units of Measurement: Inches, Millimeters
- Type of Thread: Internal, External
- Thread Specification: 1/2"-20
- Diameter of Thread: 0.5
- Length and Direction: -1
- Taper Angle: 0
- Lead of Thread: 0.05
- Threads Per Inch: 20
- Depth Per Pass: 0.003
- Number of Passes: 10
- Multistart Threads: (value: 2)
- Break Sharp Edges Pass:
- Thread Chasing Cleanup Pass:
- Plunge Angle Mode:
- Plunge Angle: 29
- Major Diameter: 0.5
- Minor Diameter: 0.4404
- Depth of Thread: 0.0298

The G-code generated in the text area is:

```
#THREADINGSUB
G91 /Use Relative Offsets

/ Setup Initial Parameters (adjustable as
necessary)
G28 (TM_Length_Of_Thread) = -1 / Z
Parameter
G28 (TM_Depth_Per_Pass) = -0.003 / I
Parameter
G28 (TM_Width_Per_Pass) = -0.001663 /
Z offset per pass
G28 (TM_Lead_Of_Thread) = 0.05 / K
Parameter
G28 (TM_Loop_Counter) = 0

/ Call the Single-Thread subroutine
G25 #SingleThreadPass L10

/ Return to starting position before the
incremental offsets.
```

Wizards- Speeds and Feeds

Opens the Cutting Speeds and Feeds Window which allows the user to calculate the spindle speed that they need to be running their machine at depending on material, number of teeth, and other variables which come into play when determining cutting speed. Simply input the proper data and let the program calculate your spindle rpm for you.

Cutting Speeds and Feeds

Compute Speeds_Feeds for High-Speed Steel Tools

Diameter of Workpiece : Inches Millimeters

Material :

Roughing Cut Finishing Cut

Cutting Speeds (sfm) :

Recommended Cutting Speed :

Cutting Speed (Feet Per Minute) : 0

Spindle Speeds (rpm) :

Recommended Spindle Speed :

Actual Spindle Speed (RPM) : 0

Recommended Feed Rates (ipr and ipm):

Feed Rate (Inches Per Revolution) :

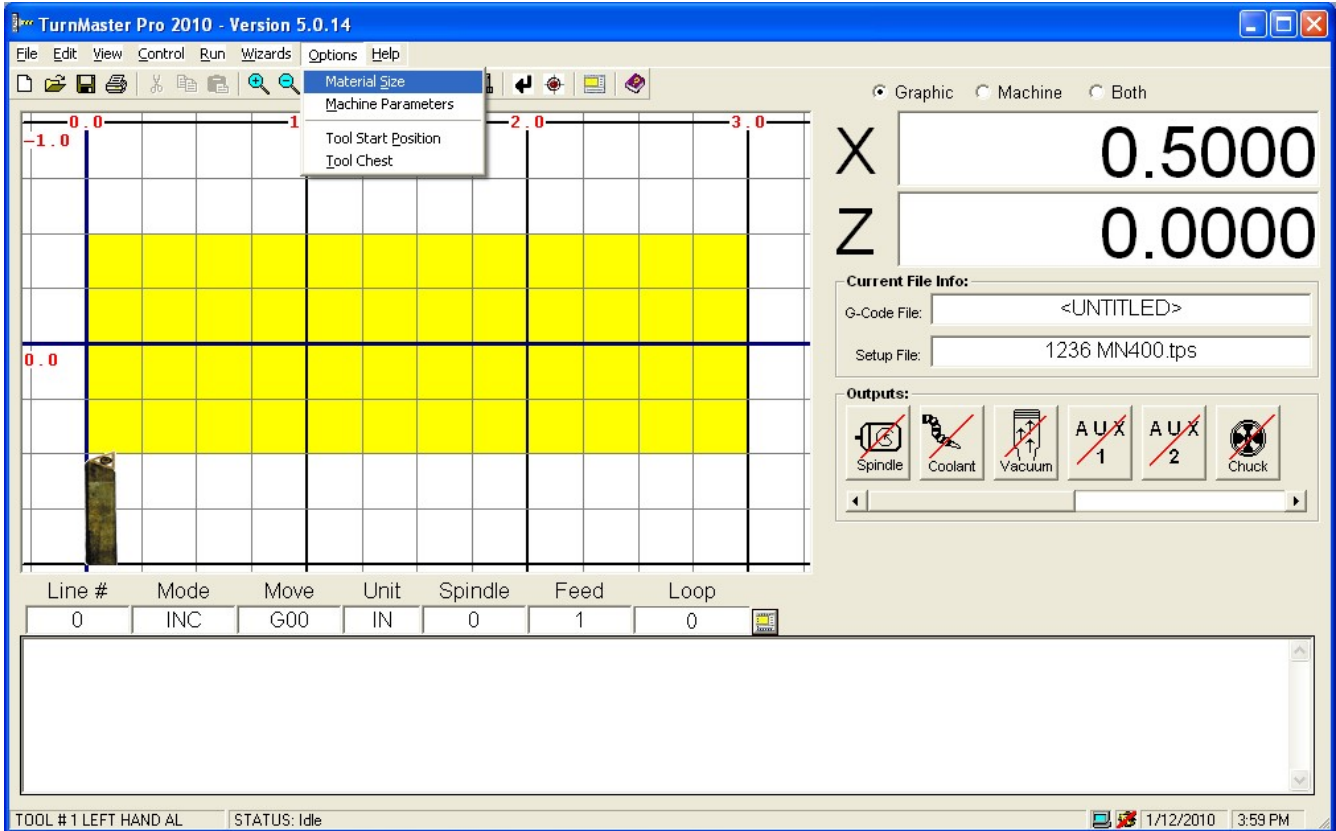
Feed Rate (Inches Per Minute) :

These Speeds/Feeds are suggestions only. Always reference the Machinery's Handbook or other published Materials. Actual applications may vary.

The **Options Menu** is used to specify the material size and to select all the options in order to properly set your machine up..

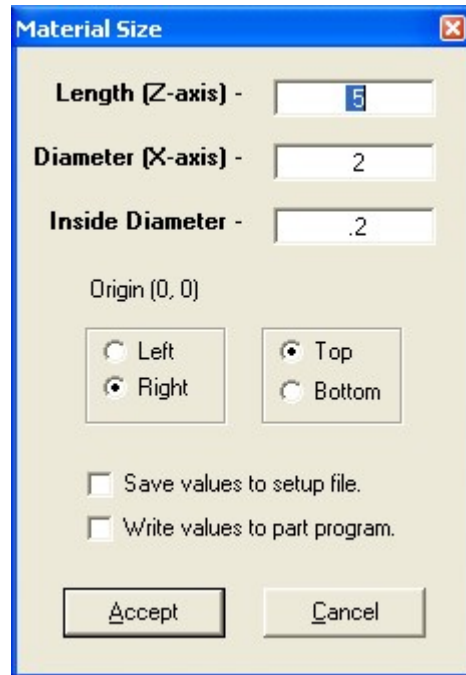
The underlined letters are called **QuickSelect** keys.

These allow access to a command only while the pull-down menu is displayed.



Options- *Material Size*

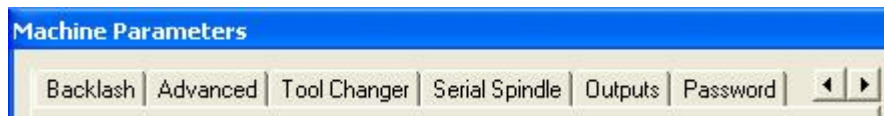
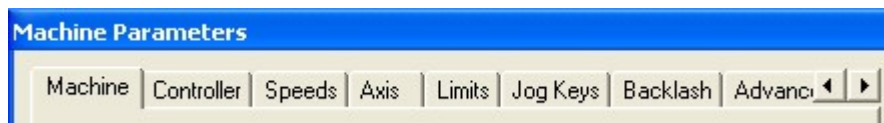
Allows the user to enter the material **Length**, **Diameter**, and/or **Inside Diameter** in the units specified in the **Machine Parameters** window.



Options- *Machine Parameters*

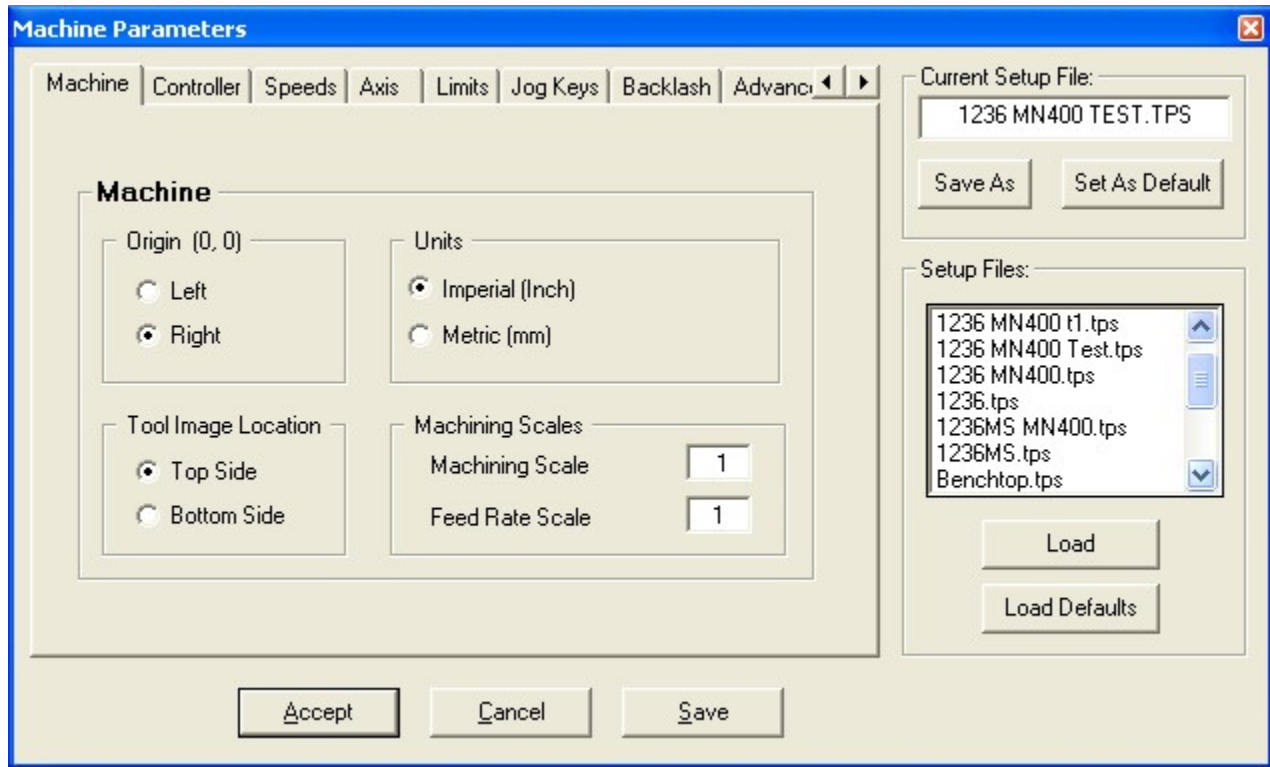
Allows the user to set Machine Parameters such as Units of Measure, Origin, select the type of controller (**MN400**, **OptiStep Plus** or **QuickPhase**), set the base address, in decimal, of the controller, Set Machine Control Speeds, Axis Control Parameters, Axis Limit Sensors, Jog Key Assignments, output delays and allows the user to setup a password field to access or change all of these parameters. The user may also select a Machine Type. This specifies what kind of machine **TurnMaster Pro** is working with. This menu can be quickly accessed with a **Mouse** click by left clicking in the whitespace of the "Setup File" box in the Current File Info. frame.

Below are two images representing the file tabs on the machine parameters page.



Options– *Machine Parameters - Machine*

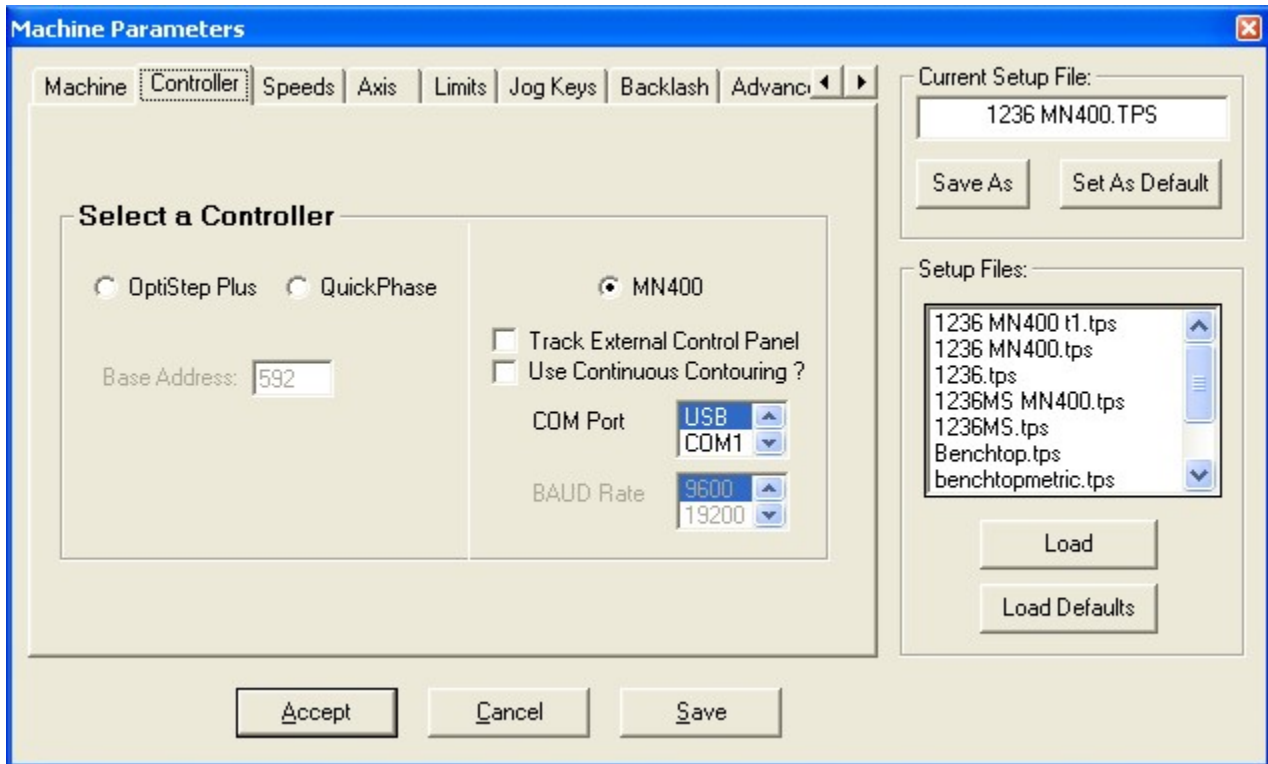
Allows the user to set the origin of the machine, unit of measurement and machine type.



Options– *Machine Parameters - Controller*

Allows the user to select which controller card they are using and enter its' corresponding base address. Also allows the user to select the COM/USB port and BAUD Rate of their connection to the MN400. You can also turn on and off the **Continuous Contouring**, and **Track External Control Panel** features found on the MN400 control card.

****NOTE:** For the OptiStep Plus and QuickPhase controllers, the DIP switch settings on the card must match the Base Address specified.



Options– **Machine Parameters - Speeds**

Allows the user to enter the rates at which the tool will travel.

Max Unramped is the maximum instantaneous rate the tool can move reliably.

Rapid Traverse is the top speed the machine can travel at with acceleration. Used for all G00 moves.

Ramp Speed is the acceleration of the stepping motors.

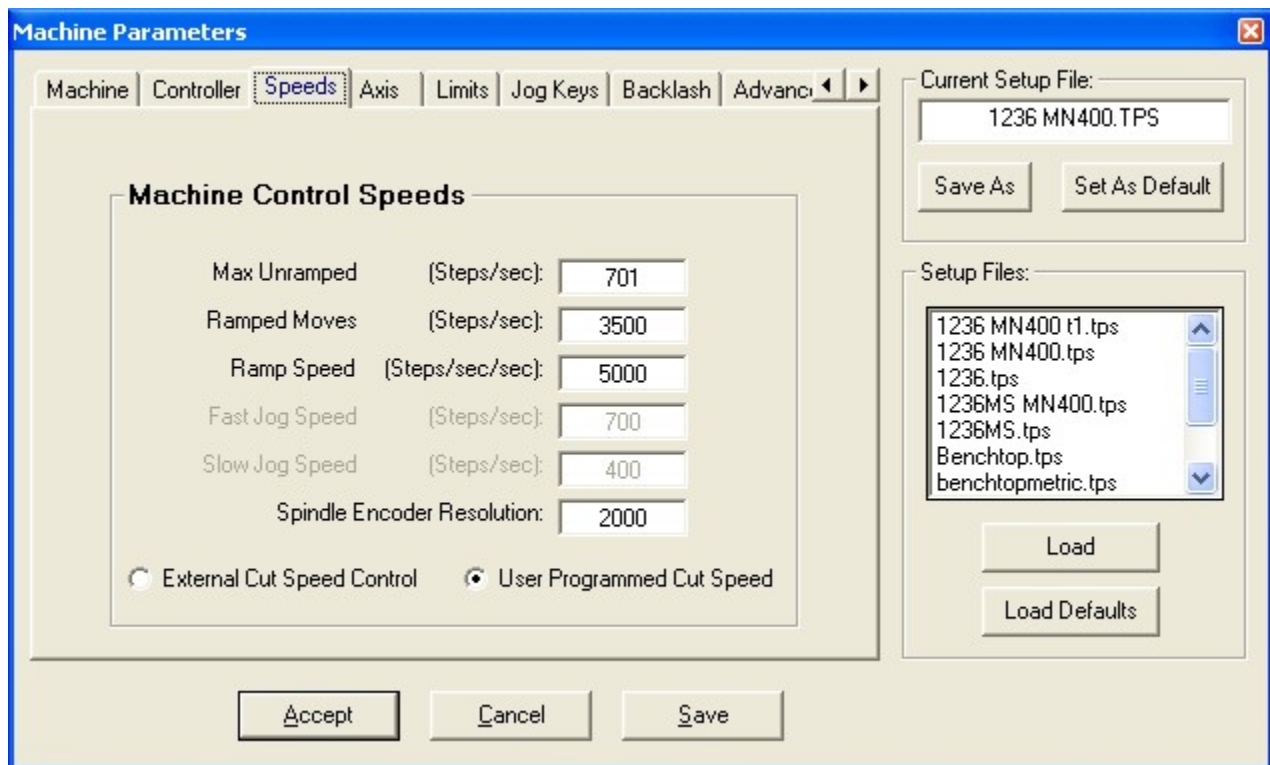
Fast Jog Speed is the rate the machine will travel in the *Jog Mode* after the “Fast Jog” button is pressed on the screen.

Slow Jog Speed is the rate the machine will travel in the *Jog Mode* after the “Slow Jog” button is pressed on the screen.

The radio buttons next to User Programmed Cut Speed or External Cut Speed Control (**for use with an OptiStep Plus or MN400 controller only**) indicates which cut speed mode is enabled by placing a black dot inside the circle. User Programmed Cut Speed mode uses the feedrate (**F**) codes embedded in the part program to set the cutting speed and External Cut Speed Control mode uses an external pulse to regulate the cutting speed.

Rapid Traverse moves are not affected by this option.

See the wiring diagram in Appendix C for instructions on how to hook-up the **OptiStep Plus** for the External Cut Speed Control option.



Options- Machine Parameters - Axis Control Parameters

Allows the user to enter the parameters that describe the lead screw pitch, the axis polarity, and the number of logical steps per stepper motor revolution. NORMAL axis polarity is defined as when the stepper motor rotates CCW, as viewed from the motor end, it generates a positive linear move. REVERSE axis polarity generates a negative linear move under the same circumstances.

****Note:** If REVERSE is selected for any axis, then the limit sensor connections must also be reversed **

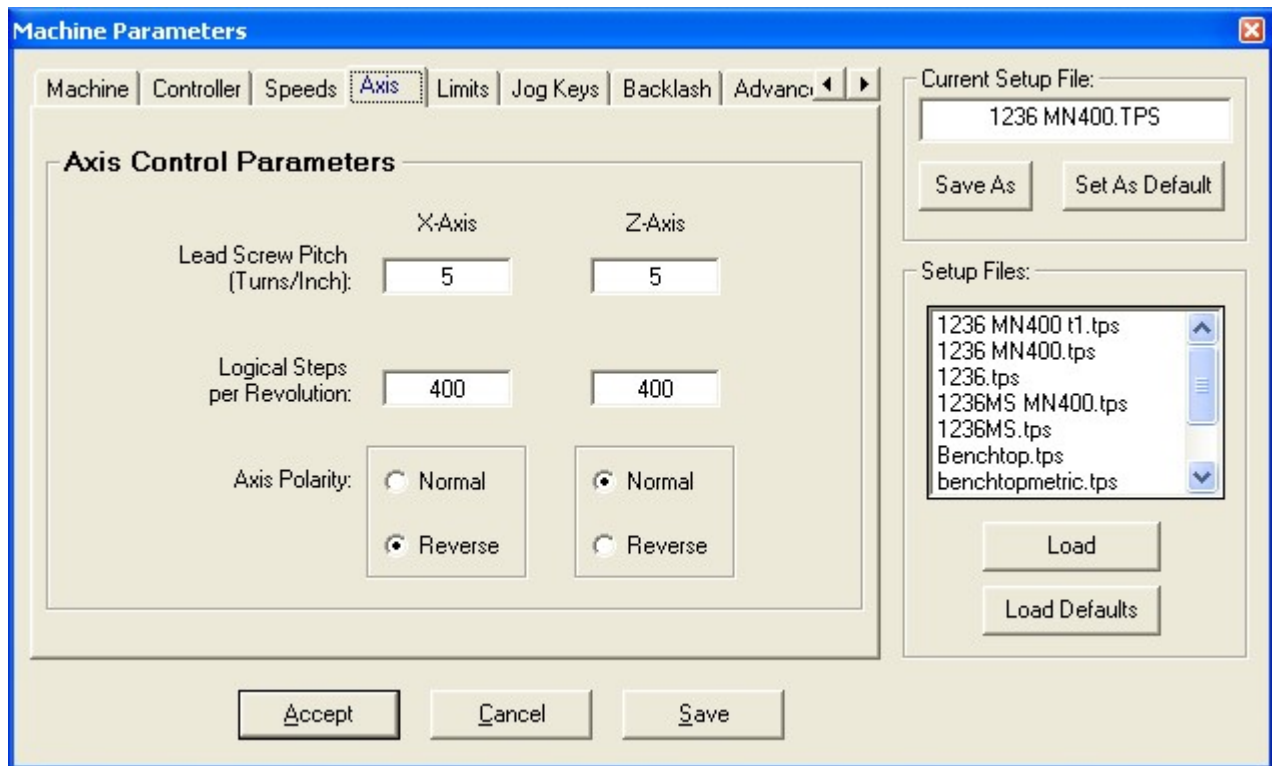
(i.e. connect a positive limit switch to the negative input).

Logical step per revolution is the number of step pulses required to move the stepping motor one full revolution taking into account the motor driver resolution setting.

Example:

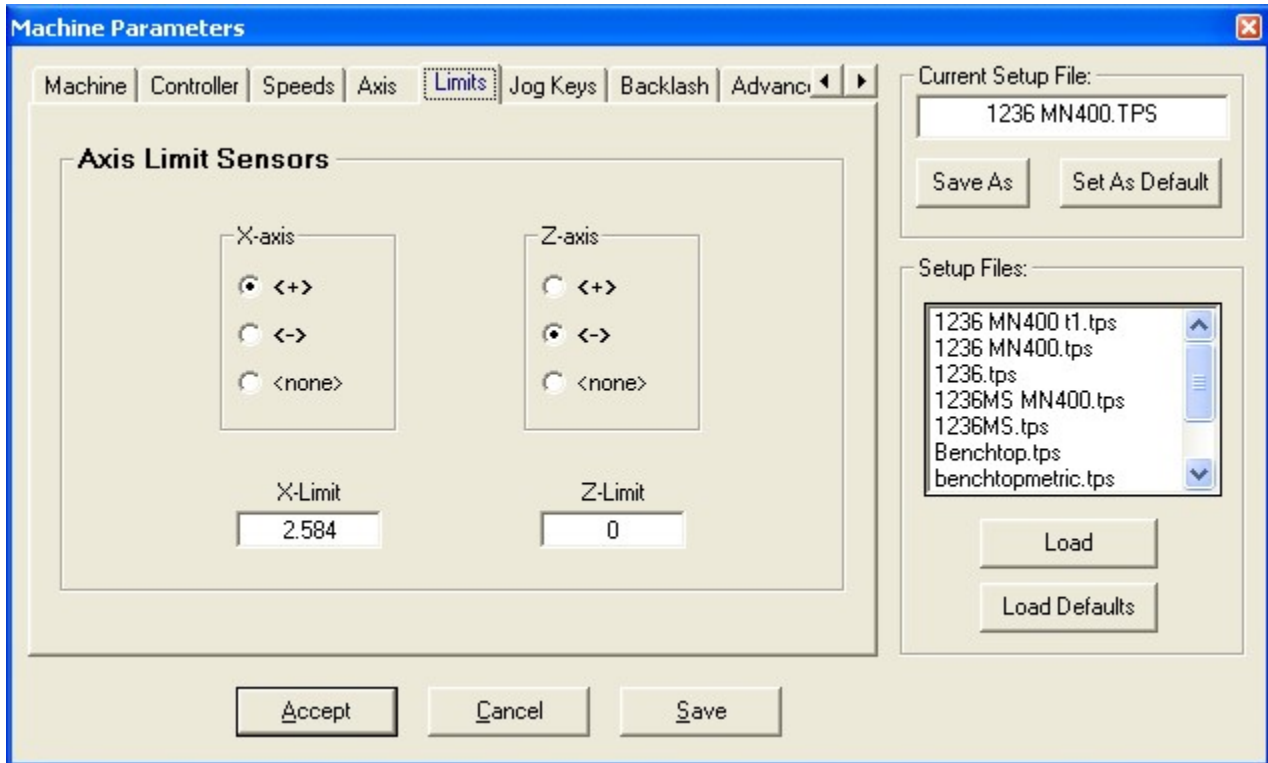
Motor Res.Driver Res.Logical Steps/Rev

200	FULLSTEP	200
200	HALFSTEP	400



Options- Machine Parameters - Limit Sensors

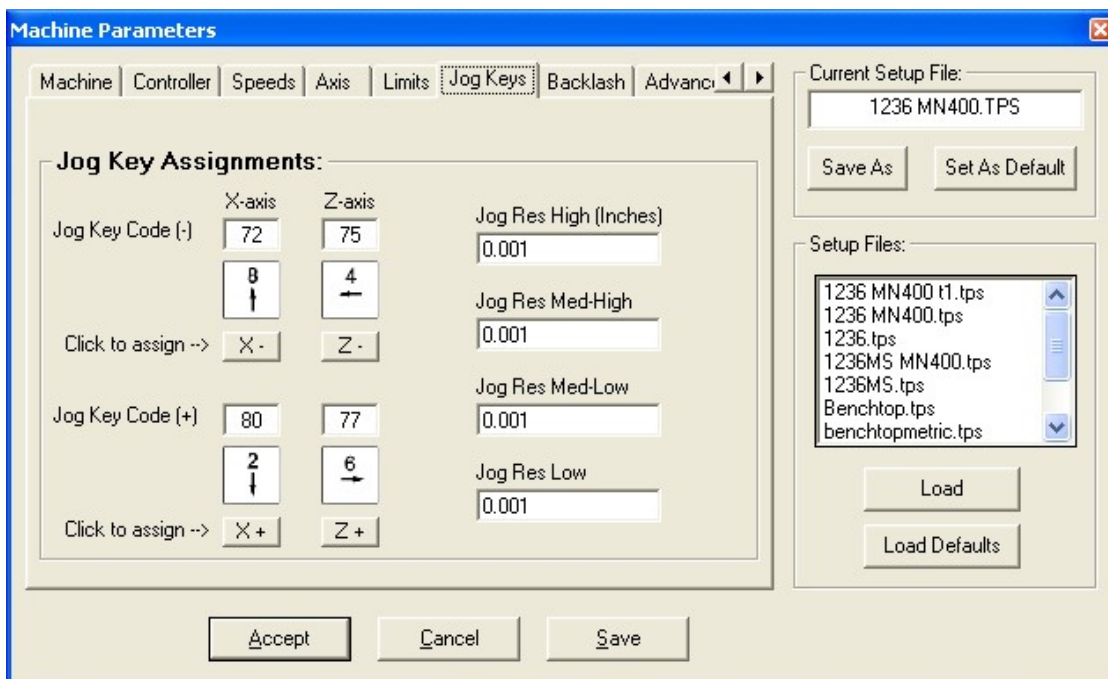
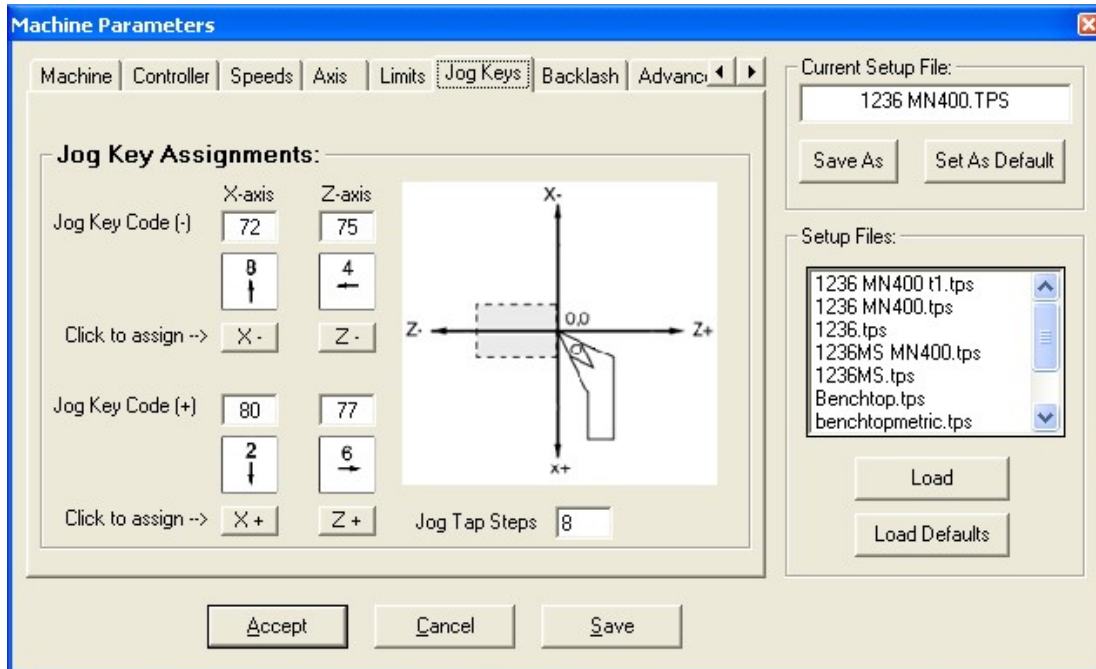
Allows the user to select the limit switches that will be used to home to. Enter the position of the home limit sensor for each axis in reference to the origin of the part. On the Z-axis only, enter the position of the bottom of the tool holder in reference to the origin of the part.



Options- *Machine Parameters - Jog Keys Assignment*

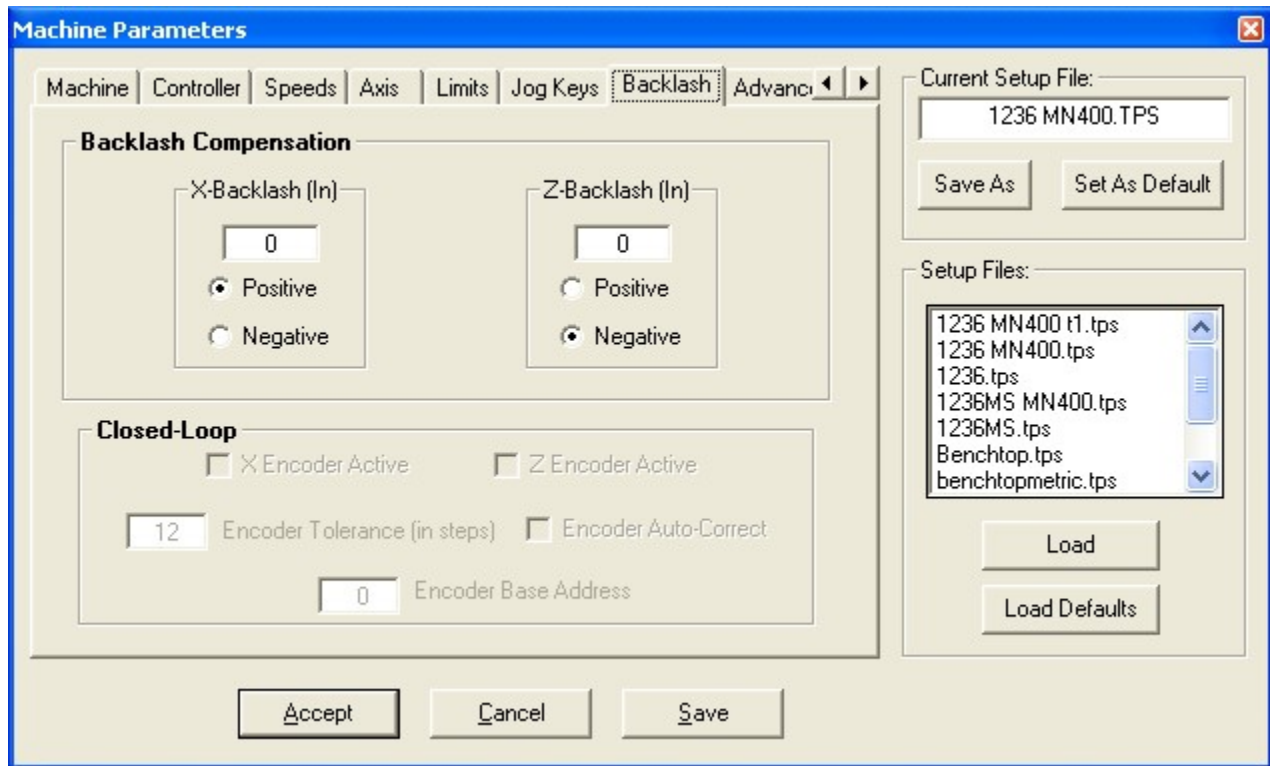
Allows the user to choose which keys on the computer keyboard control the jog movement of the machine. The default sets the Z-axis to jog with the left and right arrow keys, and the X-axis to jog with the up and down arrow keys. If you wish to change the jog keys, you should click on the button that corresponds with the axis you wish to change. This will display a small window with two buttons on it, OK and CANCEL. Press the key you wish to assign to that axis and the display window will tell you if it is a valid key. If it is, an image of the key you pressed will appear in the window. If not, a message stating 'Invalid Key' will be displayed and you will need to choose another key. To accept a valid key, click the OK button and you will notice the corresponding SCAN code is entered automatically into the Jog Keys page along with an image of that key.

You can also specify the number of steps that the motor will turn when a jog key is tapped (1-8 is the range).



Options– *Machine Parameters - Backlash compensation*

Allows the user to enter values for **X** and **Z** backlash compensation values.



Backlash is play or looseness between machine parts. If there is backlash, some commanded rotation of the motor is needed to take up this backlash before the machine starts to move.

Determine the amount of backlash in inches and select positive or negative initial compensation.

With a positive backlash setting, additional motor rotation is provided for the first negative move.

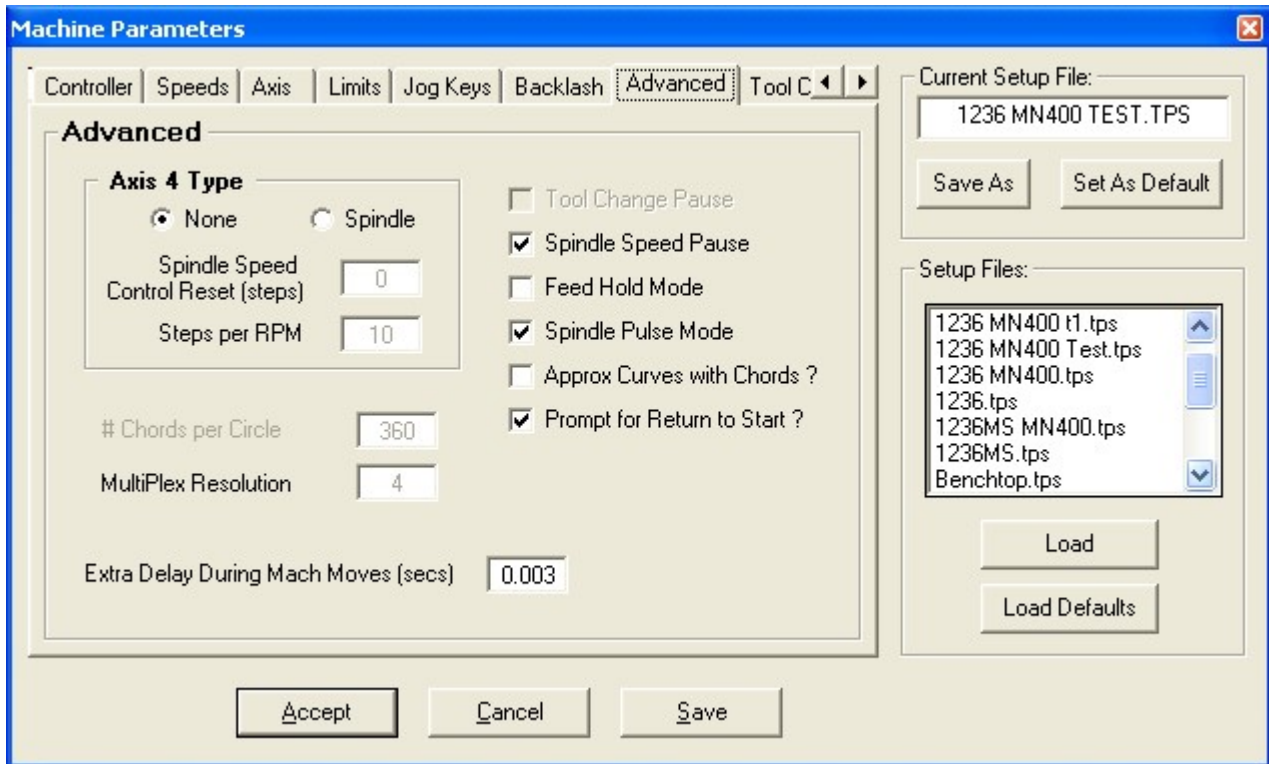
With a negative backlash setting, additional motor rotation is provided for the first positive move.

In both cases additional motor rotation is commanded at each subsequent direction reversal.

Note: After the introduction or change of a backlash setting, it is necessary to exit and restart the control software.

Options– *Machine Parameters - Advanced*

Allows the user to enter Axis-4 and other miscellaneous machine parameters.



If you select the **Spindle** option for **Axis 4 Type**, then you will need to enter a value for RPM in this field as well as the reset speed.

MultiPlex Resolution refers to resolution selection for diagonal linear moves. It allows for smoother movements while sacrificing your top speed. For most applications a multiplex of 2 works as a good compromise. 4 causes an added burden on the CPU but creates very smooth diagonal moves. 1 is for no multiplexing at all and can be a bit rough on large router tables and milling machines that do not have microstepping drivers. This value is disabled for MN400 Controllers.

Tool Change Pause - This parameter specifies the action **TurnMaster Pro** will take when it encounters an M06 command in a CNC part program. If checked, **TurnMaster** will pause the CNC program and machining so the machinist can manually change the tool. If it is not checked, the software will assume you have a multi-tool head and continue without any interruption.

Feed Hold Mode - This parameter specifies the action **TurnMaster Pro** will take when the shield is opened. If checked, the software will simply pause the machining (spindle motor remains on). If unchecked, the software will stop machining, turn the spindle motor off and prompt the user to Abort or Continue.

Spindle Pulse Mode - If checked, this parameter allows magnetic starters and 2-coil latching relays to be activated correctly while maintaining full manual control.

Spindle Speed Pause - This parameter specifies the action **TurnMaster Pro** will take when it encounters an M03 command in a CNC part program. If checked, TurnMaster will pause the CNC program and machining so the machinist can adjust the spindle speed. If it is not checked, the software will continue without any interruption.

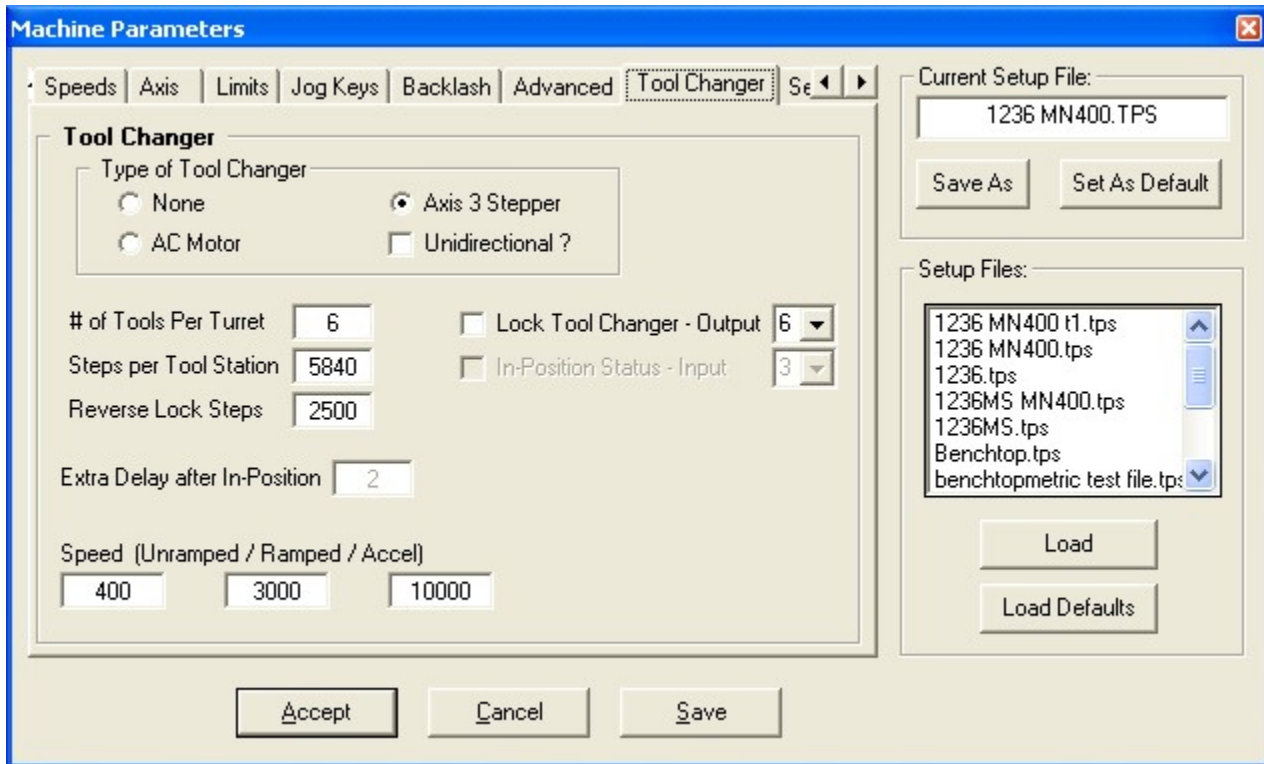
Approximate Curves with Chords ? - If checked, **TurnMaster Pro** will mill arcs as a series of straight lines. The number of straight lines (chords) per circle is determined by user input. The default is 360 chords per circle, or 1 degree per chord. Thus, using the default values, a complete circle would be milled as 360 straight lines. Note that this option only affects milling and does not affect the **Graphics** mode.

Prompt for Return to Start ? - If checked, and the current tool position is not the default start position, **TurnMaster Pro** will ask the user if he/she wants to return the tool to the start position before executing a program. If this value is unchecked, **TurnMaster Pro** will not ask the user.

Extra Delay During Machine Moves - This parameter is used to slow down queries to the controller when waiting for motion to stop after a machine move. The value must be between 0 and 1 second. The default value is 0.003 secs.

Options– *Machine Parameters - Tool Changer*

Allows the user to enter the values associated with any Tool Changer.



If you select the **Axis 3 Stepper** for **Tool Changer**, then you will need to enter the number of tools per turret, steps per tool station, whether it's a **Unidirectional Turret**, and whether to lock a separate output. You can also enter separate values for Axis 3 speeds (Unramped, Ramped, and Acceleration).

For an **AC Motor** tool changer, you will also have to select the Output # for **Tool Change Indexing**, and Input # for the **In-Position Status** signal.

Options– *Machine Parameters - Serial Spindle*

Allows the user to enter communications settings if a serial port spindle is used.

The screenshot shows the 'Machine Parameters' dialog box with the 'Serial Spindle' tab selected. The 'Serial Spindle Control' section has three radio buttons: 'None', 'FluxMaster100', and 'Teco 7300 CV', with 'Teco 7300 CV' selected. Below this is a note: 'If you are unable to communicate with the VFD, select Com Port from the dropdown list and click <Get Settings>.' The 'Com Port' dropdown is set to '1', 'Baud Rate' to '9600', and 'Com Settings' to 'N,8,1'. A 'Recommended Settings' section shows '9600' and 'N,8,1' with a 'Get Settings' button. Other settings include 'Threading Start Speed' (100 RPM), 'Accel/Decel Time' (3 Sec), and 'Gear Selection' (780 RPM). A 'Write Default VFD Settings' button is also present. On the right, the 'Current Setup File' is '1236 MN400 TEST.TPS' with 'Save As' and 'Set As Default' buttons. The 'Setup Files' list includes '1236 MN400 t1.tps', '1236 MN400 Test.tps', '1236 MN400.tps', '1236.tps', '1236MS MN400.tps', '1236MS.tps', and 'Benchtop.tps', with 'Load' and 'Load Defaults' buttons below. At the bottom are 'Accept', 'Cancel', and 'Save' buttons.

Options– *Machine Parameters - Outputs*

Allows the user to enter specific delay values for up to eight numbered outputs.

The screenshot shows the 'Machine Parameters' dialog box with the 'Outputs' tab selected. The dialog has a title bar with a close button. Below the title bar is a tabbed interface with tabs for 'Limits', 'Jog Keys', 'Backlash', 'Advanced', 'Serial Spindle', 'Outputs', and 'Password'. The 'Outputs' tab is active and contains a table with 8 rows. Each row has three input fields: 'Name', 'Delay After On', and 'Delay After Off'. The first six rows are filled with 'test1' through 'test6', and the last two are empty. The 'Delay After On' values are 0, 0, 1, 0, 0, 0, 0, 0. The 'Delay After Off' values are 0, 0, 2, 0, 0, 0, 0, 0. To the right of the table is a 'Current Setup File' section with a text box containing '1236 MN400.TPS' and 'Save As' and 'Set As Default' buttons. Below that is a 'Setup Files' section with a list box containing several files: '1236 MN400 t1.tps', '1236 MN400.tps', '1236.tps', '1236MS MN400.tps', '1236MS.tps', 'Benchtop.tps', and 'benchtopmetric.tps'. There are 'Load' and 'Load Defaults' buttons below the list. At the bottom of the dialog are 'Accept', 'Cancel', and 'Save' buttons.

	Name:	Delay After On:	Delay After Off:
1	test1	0	0
2	test2	0	0
3	test3	1	2
4	test4	0	0
5	test5	0	0
6	test6	0	0
7		0	0
8		0	0

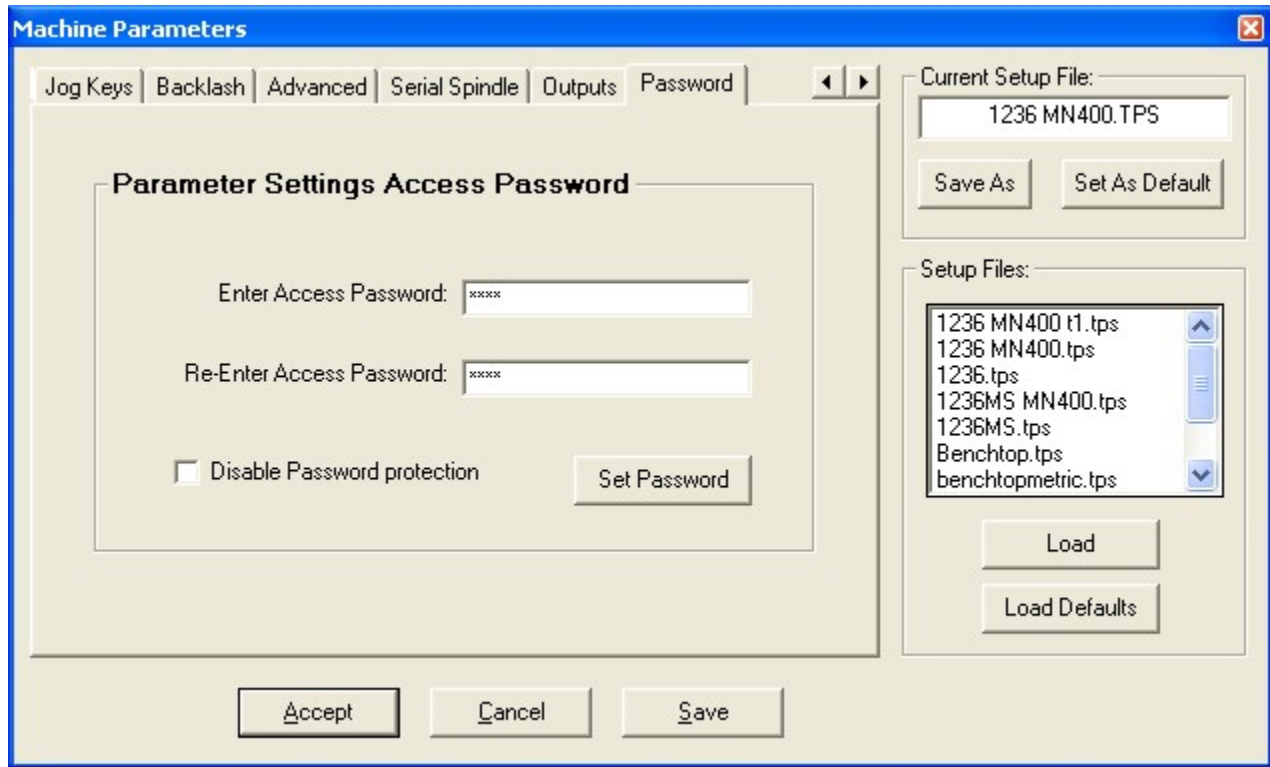
Current Setup File: 1236 MN400.TPS

Setup Files:

- 1236 MN400 t1.tps
- 1236 MN400.tps
- 1236.tps
- 1236MS MN400.tps
- 1236MS.tps
- Benchtop.tps
- benchtopmetric.tps

Options- Machine Parameters - Password

Allows the user to specify a password to protect the machine parameters page from unauthorized changes that could cause damage to the machine and/or cause injury.



Options- Machine Parameters - Save As

Allows the user to save the current setup parameters to a setup file of choice or create a new one.

Options- Machine Parameters - Set As Default

Saves the current settings to the default configuration file. All the options set will become permanent.

Options- Machine Parameters - Load

Allows the user to load a setup file. A setup file has a .TPS filename extension and configures **TurnMaster Pro** with the setup parameters in the file. This allows the user to quickly change between different setups.

Options- Machine Parameters - Load Defaults

Clears the current settings and restores the defaults from disk.

Options- *Machine Parameters - Accept*

The current setup parameters will be used for this session but they have NOT been saved to the setup file.

Options- *Machine Parameters - Cancel*

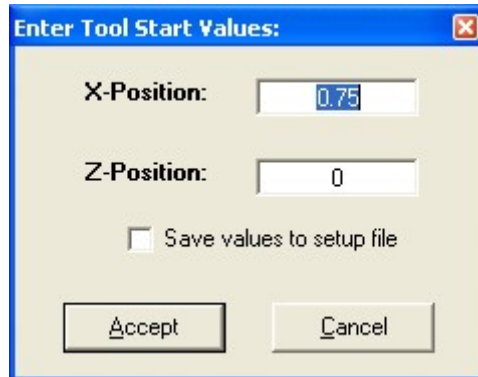
Cancel any changes made to the current setup parameters.

Options- *Machine Parameters - Save*

Saves the current setup parameters to the current setup file.

Options- *Start Position*

Allows the user to specify the position of the tool at the beginning of the program.



Options- Tool Chest

Opens the Tool Chest Window which allows the user to describe the **X, Z offset** and **tool nose radius** of each tool. This screen also allows you to associate an image file with your tool.

You can use the scroll bar to find the desired tool to change or find an empty tool. **Turnmaster Pro** allows up to 99 tools.

TurnMaster Pro comes with a set of tool images, which are contained in the TMPFW directory on your hard drive, usually under Program Files. After associating an image file, you will see it appear in the image box when you select the tool in the list on the left hand side of the window.

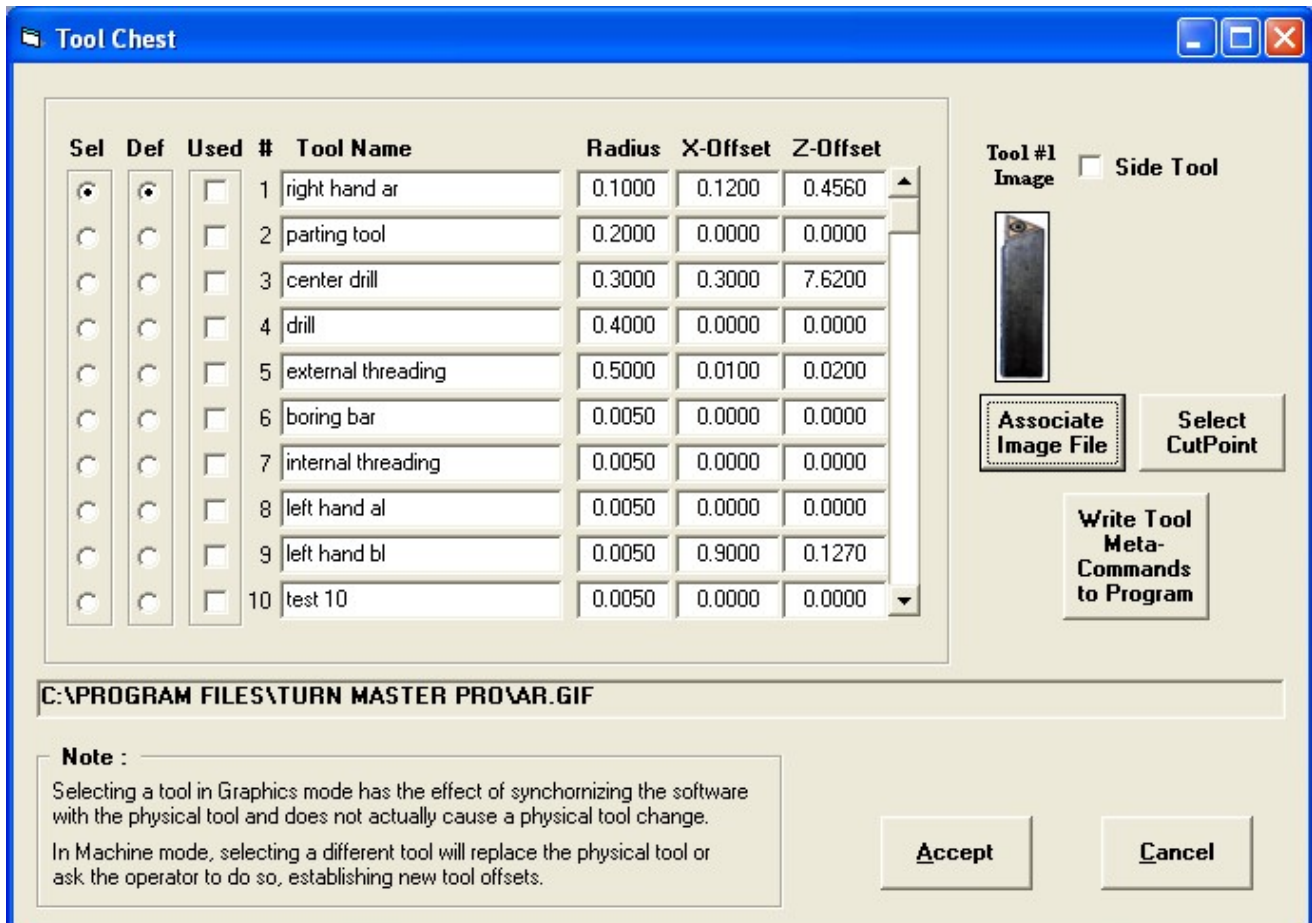
You will also notice a checkbox to the right of the tool image box that says '**Side Tool**'. By clicking this button you will see the tool image box reposition itself in a horizontal format. This is for proper depiction of side tools, such as a boring bar.

The Tool Nose Radius entry on this window corresponds to the width of the cut in the viewport on the main screen. This parameter is for graphical purposes only and will not affect your part. If the cut lines in the viewport are too thick or too thin, this parameter is the one to adjust.

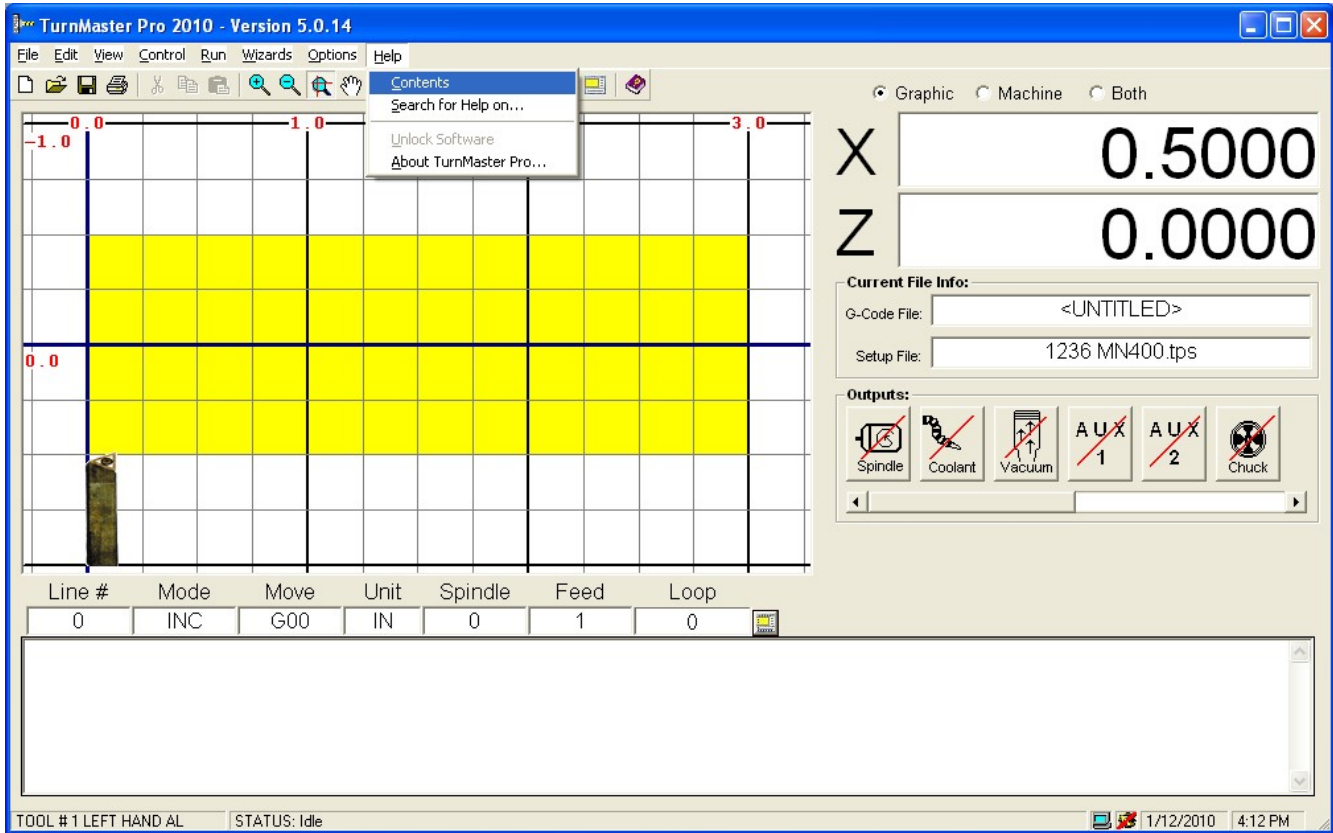
You can change the default tool (tool selected when starting **TurnMaster Pro**) through this window by clicking the **Def** (Default) option button for desired tool.

You can also change the currently selected tool by clicking the **Sel** (Selected) option button for the desired tool. The 15th icon in the Toolbar also corresponds to this menu command.

****Note:** There are 2 picture files for each tool number. For Example, tool1.gif and tool1x.gif. The image files that contain the **x** in there names are used when **Top Side Tool** location is selected under Machine Parameters. The image files without the **x** are used for **Bottom Side Tool** location, which is the default.



The Help Menu.



Help- Contents

This will display a the Table of Contents from the .PDF Help file. The right-most icon in the Toolbar also corresponds to this menu command

Help- Search For Help On

This will display the first page the .PDF Help file where you can begin your search.

Help- Unlock Software

This will display the Unlock Software window, where, when purchased, allows you to enter an Unlock Code to fully maximize the use of the software. Having an unlocked copy allows you to utilize Tech Support for mechanical and software related problems.

****NOTE:** Once the software is unlocked, the Unlock Software option on the help menu will be disabled permanently.

Help- About

This will display the currently running version of **TurnMaster Pro**, Instep or MN400 Library, Company, and Copyright information.



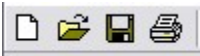
Toolbar and Icon Related Commands

This section will explain to and instruct you on the Toolbar and Icon related commands.

The graphic below is the Toolbar at program start-up.



The following are short descriptions of the above graphical commands contained on the toolbar (from left to right).

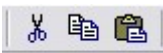


New - Clears the part program in memory and establishes the standard defaults for a new part program.

Open - Displays the Open File Dialog Window which displays all part programs in the current directory.

Save - Quickly saves the current part program to disk. The extension CNC is automatically assigned

Print - Displays the Print Dialog Window, which allows the user to choose Print Options, and utilize the built-in Print Preview Function.



****Notice** that the second set of icons in the top Toolbar graphic appear "grayed" out and in the above Toolbar they are active. These are the Edit Functions that are available only during CNC Edit Mode.

Cut - Cuts the currently highlighted text out of the page and onto the clipboard.

Copy - Copies the currently highlighted text onto the clipboard.

Paste - Inserts the current data from the clipboard to the current cursor location.



Zoom In - Zooms the current cutting window in by 25%

Zoom Out - Zooms the current cutting window out by 25%

Zoom - Zooms to the limits specified by the last mouse drag

Pan - Pans the current cutting window. When in **Pan** mode, the screen cursor will appear as a hand. To exit **Pan** mode, press the **Pan Icon**, or any of the three **Zoom** icons.



Start/Stop Execution - Executes the current part program in memory in continuous mode. After clicking the **GO icon** you will notice that it changes to a **STOP icon**. This allows you to **STOP** the program execution at any time. Either **Graphics Mode** or **Machine Mode** must be enabled to use this option. Alternatively, you can press any key on the keyboard to pause the program.

Pause Execution - Pauses program execution and displays **Step/Auto/Continue window**. Pressing any key during program execution also brings up this window. If you are in the middle of a G02 or G03 command, the program will not pause until after the command is completed.

Step Mode - Starts currently loaded program in **Step mode**, displaying the **Step/Auto/Continue window**.

Tool Chest - Opens the Tool Chest Window which allows the user to describe the length and radius of each tool. When changing tools, Z-axis compensation will occur automatically based on the difference in tool lengths.

Return Tool to Start - Returns the tool to the starting position.

Init as Start Position - Initiates the current position of the tool as the Start Position and resets the internal counters. The current location of the tool is assumed to be in the tool start position.

CNC Full-Screen Edit Mode - Toggles between full-screen CNC edit mode and graphics mode.

Note: This icon is also displayed above the small **CNC Edit** textbox, to the right of the status fields. This allows easy access to the Full-Screen Edit Mode when typing in the small **CNC Edit** textbox.

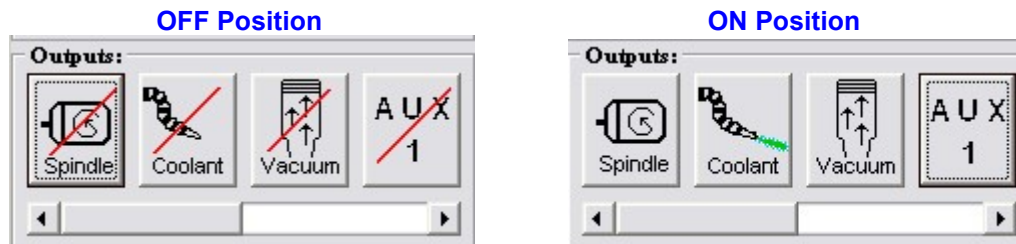


Help - Displays a window showing the contents of this help file.

ICON RELATED COMMANDS

Along with the Toolbar, there are other graphical/icon related command keys.

Below are pictures of the Outputs graphical menu in the **OFF** position and the **ON** position.



These graphical Icons/command buttons control all outputs. The following are short descriptions of the above graphical commands contained on the Outputs graphical menu. (from left to right)

Spindle - Toggles **On/Off** the **spindle** of the milling machine.

Coolant - Toggles **On/Off** the flow of **coolant (if applicable)** for the milling machine. If the machine type is set to TORCH, this icon would be replaced with a torch icon and work in the same fashion.

Vacuum - Toggles **On/Off** the power to the **vacuum (if applicable)** of the milling machine.

AUX 1 - Toggles **On/Off** the **auxiliary 1 output (if applicable)**

AUX 2 - Toggle **On/Off** the **auxiliary 2 output (if applicable)**

AUX 3 - Toggles **On/Off** the **auxiliary 3 output (if applicable)**

AUX 4 - Toggle **On/Off** the **auxiliary 4 output (if applicable)**

Chuck - Toggles **Open/Close** the **pneumatic chuck (if applicable)** of the milling machine

****NOTE:** The Red Slash across the button indicates that Output is OFF.

****NOTE:** These outputs will function only while in machine mode.

Function Detail Reference

Preparatory Functions (G-Codes):

G00

This function is used to rapidly locate the tool to a new location.
This should only be used when **no contact** with the material is taking place.

EXAMPLE: G00 X0.1 Z-0.1

G01

This code is used to cut along the path of a straight line.

The first G01 in a program will require an **F** (feedrate) parameter unless you want to use the feedrate already loaded into memory. You may change the feedrate in the same manner whenever you have a G01 command. **F** Codes are the **Feedrate** in inches/minute multiplied by 10 when in **Inch** mode. For example **F15** would equate to 1.5 inches/minute. In **Metric** mode **F** codes are in mm/minute.

EXAMPLE: G01 X1.2 Z0.1 F20 / Feedrate is 2.0 inches/minute

G02

This is used for a **two-axis** circular cutting move in a **clockwise** direction (as viewed from the TOP half of the graphical part).

To perform a circular move the cutting tool is moved to the **starting position** before the **G02 code** is used. Following the **G02 code**, the end point of the arc is expressed in **two axes (X and Z)**. Next, the distance from the **start of the arc** to the **circle center point** is given with an **I and/or K specification**. At least one of **I** and **K** must be specified. When an **I** or **K** is not specified, it is assigned to be **zero**.

You may use the **R** (Radius) parameter as an alternative to using the **I** and **K** parameters. **TurnMaster** will compute up to 2 possible center points with the specified start and end points and the given radius. If the **R** parameter is positive, the center point that results in the shortest arc will be selected. If the **R** parameter is negative, the center point that results in the longest arc will be selected.

EXAMPLES: G02 X.5 Z2 K1 F10
 G02 X.5 Z2 R1 / Draw shortest arc of Radius 1
 G02 X.5 Z2 R-1 / Draw longest arc of Radius 1

G02 (Continued)

DETAILED EXAMPLE #1:

G75 / Select multiple quadrant mode
G90 / Set absolute programming mode
G00 X0.5 Z0.0 / Position at (x, z) = (0.5, 0)
G02 X0.5 Z-0.8 R1 / Draw Clockwise Arc - Endpoint at (x, z) = (0.5, -0.8)
/ Right to Left, Radius of circle = 1

The screenshot displays the TurnMaster Pro 2012 software interface. The main window shows a coordinate grid with X and Z axes. The X-axis ranges from -2.0 to 0.0, and the Z-axis ranges from 0.0 to -0.8. A yellow shaded area represents the workpiece. A pink arc is drawn from X=0.5, Z=0.0 to X=0.5, Z=-0.8, representing the G02 command. The software title bar indicates 'TurnMaster Pro 2012 - Version 5.2.124'. The menu bar includes File, Edit, View, Control, Run, Wizards, Options, and Help. The toolbar contains various icons for file operations and machine control. The right panel shows the current file info: G-Code File: G02 G03 DOC SAMPLES.CNC, Setup File: 1236 MN400.tps. The outputs section includes Spindle, Coolant, Vacuum, AUX 1, AUX 2, and Chuck, all of which are currently disabled. The bottom status bar shows 'TOOL # 5 EXTERNAL THREAD' and 'STATUS: Idle'. The system tray at the bottom right shows the date '5/3/2012' and time '9:33 AM'.

Line #	Mode	Move	Unit	Spindle	Feed	Loop
6	ABS	G02	IN	0		0

```
G75 / Select multiple quadrant mode
G90 / Set absolute programming mode
G00 X0.5 Z0.0 / Position at (x, z) = (0.5, 0)
G02 X0.5 Z-0.8 R1 / Draw Clockwise Arc - Endpoint at (x, z) = (0.5, -0.8)
/ Right to Left, Radius of circle = 1
```

TOOL # 5 EXTERNAL THREAD | STATUS: Idle

5/3/2012 9:33 AM

G02 (Continued)

DETAILED EXAMPLE #2:

G75 / Select multiple quadrant mode
G90 / Set absolute programming mode
G00 X0.5 Z-2 / Position at (x, z) = (0.5, -2)
G02 X0.5 Z-1.2 R1 / Draw Clockwise Arc - Endpoint at (x, z) = (0.5, -1.2)
/ Left to Right, Radius of circle = 1

The screenshot displays the TurnMaster Pro 2012 software interface. The main window shows a 2D coordinate system with X and Z axes. The X-axis ranges from -2.0 to 0.0, and the Z-axis ranges from 0.0 to -1.2. A yellow shaded area represents the workpiece, and a pink arc is drawn from X=0.5, Z=-2.0 to X=0.5, Z=-1.2. The software title bar indicates 'TurnMaster Pro 2012 - Version 5.2.124'. The right-hand panel shows the current file information and outputs. The current file is 'G02 G03 DOC SAMPLES.CNC' and the setup file is '1236 MN400.tps'. The outputs section shows icons for Spindle, Coolant, Vacuum, AUX 1, AUX 2, and Chuck, all of which are currently disabled. The bottom status bar shows 'TOOL # 5 EXTERNAL THREAD | STATUS: Idle' and the date/time '5/3/2012 9:37 AM'.

Line #	Mode	Move	Unit	Spindle	Feed	Loop
6	ABS	G02	IN	0		0

```
G75 / Select multiple quadrant mode
G90 / Set absolute programming mode
G00 X0.5 Z-2 / Position at (x, z) = (0.5, -2)
G02 X0.5 Z-1.2 R1 / Draw Clockwise Arc - Endpoint at (x, z) = (0.5, -1.2)
/ Radius of circle = 1
```

TOOL # 5 EXTERNAL THREAD | STATUS: Idle 5/3/2012 9:37 AM

G03

This code is used for **counterclockwise** cutting moves.
It is used in the same way as the **G02 command** described above.

EXAMPLES: G00 X3 Z-.2
G03 X4 Z2.2 I.4375 F10
G03 X.5 Z2 R1 / Draw shortest arc of Radius 1
G03 X.5 Z2 R-1 / Draw longest arc of Radius 1

DETAILED EXAMPLE #1:

G75 / Select multiple quadrant mode
G90 / Set absolute programming mode
G00 X0.5 Z0.0 / Position at (x, z) = (0.5, 0)
G03 X0.5 Z-0.8 R1 / Draw Counter-Clockwise Arc - Endpoint at (x, z) = (0.5, -0.8)
/ Right to Left, Radius of circle = 1

The screenshot displays the TurnMaster Pro 2012 software interface. The main window shows a coordinate grid with X and Z axes. The X-axis ranges from -2.0 to 0.0, and the Z-axis ranges from 0.0 to -0.8. A yellow shaded area represents the workpiece. A pink arc is drawn from X=0.5, Z=0.0 to X=0.5, Z=-0.8, with a radius of 1. The software status bar at the bottom indicates "TOOL # 5 EXTERNAL THREAD" and "STATUS: Idle".

TurnMaster Pro 2012 - Version 5.2.124

File Edit View Control Run Wizards Options Help

Graphic Machine Both

X 0.5000
Z -0.8000

Current File Info:
G-Code File: G02 G03 DOC SAMPLES.CNC
Setup File: 1236 MN400.tps

Outputs:
Spindle Coolant Vacuum AUX 1 AUX 2 Chuck

Line #	Mode	Move	Unit	Spindle	Feed	Loop
5	ABS	G03	IN	0		0

G75 / Select multiple quadrant mode
G90 / Set absolute programming mode
G00 X0.5 Z0.0 / Position at (x, z) = (0.5, 0)
G03 X0.5 Z-0.8 R1 / Draw Counter-Clockwise Arc - Endpoint at (x, z) = (0.5, -0.8)
/ Radius of circle = 1

TOOL # 5 EXTERNAL THREAD | STATUS: Idle 5/3/2012 9:42 AM

G03 (Continued)

DETAILED EXAMPLE #2:

G75 / Select multiple quadrant mode
G90 / Set absolute programming mode
G00 X0.5 Z-2 / Position at (x, z) = (0, -2)
G03 X0.5 Z-1.2 R1 / Draw Counter-Clockwise Arc - Endpoint at (x, z) = (0.5, -1.2)
/ Left to Right, Radius of circle = 1

TurnMaster Pro 2012 - Version 5.2.124

File Edit View Control Run Wizards Options Help

Graphic Machine Both

X 0.5000
Z -1.2000

Current File Info:
G-Code File: G02 G03 DOC SAMPLES.CNC
Setup File: 1236 MN400.tps

Outputs:
Spindle Coolant Vacuum AUX 1 AUX 2 Chuck

Line #	Mode	Move	Unit	Spindle	Feed	Loop
6	ABS	G03	IN	0		0

```
G75 / Select multiple quadrant mode
G90 / Set absolute programming mode
G00 X0.5 Z-2 / Position at (x, z) = (0, -2)
G03 X0.5 Z-1.2 R1 / Draw Counter-Clockwise Arc - Endpoint at (x, z) = (0.5, -1.2)
/ Left to Right, Radius of circle = 1
```

TOOL # 5 EXTERNAL THREAD | STATUS: Idle 5/3/2012 9:49 AM

G04

This code causes the program to **pause** for a set period of time during execution. A **G04** followed by a **D** and the number of seconds of the delay.

EXAMPLE: G04 D1.5 / Pause for 1.5 seconds

G25

This code causes program execution to continue at the first block of a named subroutine. Certain modal states are saved automatically and restored upon return from subroutine. These include **G90/91**, **G74/75**, and current program line. An **L** can be used here to call the subroutine a specified number of times.

EXAMPLE:

G25 #GROOVE L5

Runs the **GROOVE** subroutine in the current file five times.

G25 #SUBS.GROOVE

Runs the **GROOVE** subroutine in the file **SUBS.CNC** in the current directory.

G26

Conditional Branch: this code causes the program to jump to the specified label if the specified input is a **active** (low voltage) or if the **parameter is true**. Valid input #s are **I1 - I8**.

Note: The **I**-Parameter is only available with the **OptiStep Plus** and **MN400** controllers.

If an **I** parameter is not used, the formula between the **G26** and the label is evaluated. If the result is arithmetically non-zero or logically true, then the jump will occur. If the formula is evaluated as arithmetic zero or logical false, then the jump will not occur.

Note: To use logical comparisons, use the operands =, >, <, >=, <=, and enclose the entire expression in square brackets [].

EXAMPLE 1:

```
/-- Jump to #HOLESET when input 1 is a logical zero.  
G26 I1 #HOLESET
```

EXAMPLE 2:

```
/-- If the value of (parts) is greater than or equal to 1, goto #NextPart  
G26 [(parts) >= 1] #NextPart
```

EXAMPLE 3:

```
G28 (parts) = 5  
G90       /ESTABLISH ABSOLUT  
G75  
G92 X0 Y0 Z0.5 /ESTABLISH START POINT  
# NextPart  
/program goes here  
/  
G28 (parts) = [(parts) - 1]  
/-- If the value of (parts) is greater than or equal to 1, goto #NextPart  
G26 [(parts) >= 1] #loop  
M25       /RETURN TO START POINT  
M2
```


G27

Unconditional Branch: this code causes the program to jump to the specified label.

EXAMPLE: G27 #HOLESET

Will jump to #HOLESET.

G28

This is used to set a variable to a desired value. The variable should be made up of letters and numbers only and must be enclosed in parentheses. Once the variable is defined, it may be used in any succeeding blocks in place of numerical constants.

You may also use mathematical formulas on the right side of the equal sign. If a formula is used, it must be surrounded by square brackets "[]" in order for the math evaluator to interpret it correctly. See the section "*Using the Math Formula Evaluation Feature*" for more information.

EXAMPLE: G28 (DEPTH)=.050 / Sets the DEPTH to .050
 G91 / Incremental
 G01 X-(DEPTH) F10 / Dig into material
 G00 X(DEPTH) / Rapid traverse out of material

 G28 (DEPTH) = [(DEPTH) * 2] / Set Depth to twice the current depth.

G33

Canned threading cycle. Allows internal and external, right and left handed threads to be cut. The parameters include: **X** final depth of the thread, **Z** thread length, **K** pitch of the thread, and **Q** # of passes of **I** depth per pass (see **Appendix D**).

An optional **D** (dwell) parameter can be added. If present, movement will pause for the specified time at the end of each pass. This has the effect of cutting a shallow groove at the end of the cut.

X should be 0.050" above material before command.

EXAMPLE: G33 X-.065 Z-1 K.1 Q5
 G33 X-.065 Z-1 K.1 Q5 D0.5

will cut a 10 threads per inch, 0.065" deep external right-handed thread 1" long in 5 cutting passes.

G60

Turn off **Continuous Contouring Mode**.

The checkbox on the **Machine Parameters** screen will be the initial setting of *Continuous Contouring* each time a program starts up.

G64

Turn on **Continuous Contouring Mode**.

G70

This code sets the inch programming format. This overrides the unit of measure selected in the **Material Size/Setup Window**.

EXAMPLE: G70

G71

This code sets the metric programming format. This overrides the unit of measure selected in the **Material Size/Setup Window**.

EXAMPLE: G71

G74

This code selects the **Single Quadrant Arc Rrogramming Mode**. (I & K parameters must be positive.)

EXAMPLE: G74

G75

This code selects **Multiple Quadrant arc Programming Mode**. (I & K parameters may be positive or negative.) **G75** is the default mode for **TurnMaster Pro**.

EXAMPLE: G75

G80

Cancels a **CANNED CYCLE**. Use this command after each **CANNED CYCLE function**.

G81

This is a canned outside-diameter and inside-diameter turning cycle. Its parameters are:

- Z** incremental length and direction of cut
- X** incremental depth and direction of cut
- Q** number of passes (use either **Q** or **I**)
- I** depth per pass (use **I** or **Q**)

Note: The cutting tool must be **0.05 inches (1.27 mm)** from the material in the x direction before the **G81** is executed. After **G81**, the tool returns to that position.

Here are sample programs using metric and imperial units. In both cases, the radius of the material is 0.5 inches (12.7 mm).

Imperial	Metric
G90	G90
G70	G71
G00 X0.55 Z0	G00 X13.97 Z0
G81 Z-0.5 X-0.25 I0.05 F33	G81 Z-5.5 X-1.5 I0.25 F33
This reduces the radius 0.25 inches, 0.05 inches each pass.	This reduces the radius 1.5mm, 0.25 mm each pass.

G82

Canned cycle for reducing part length. Parameters include **X**, the depth of each cut; **Z**, the distance along the **Z-axis** that will be faced; and either **Q**, the number of cuts to make, or **K**, the distance (on the **Z-axis**) that the tool will move after each cut.

Z should be 0.050" above material before command.

EXAMPLE: G82 X.55 Z.1 Q3
 G82 X.55 Z.1 K.04

Will face 0.100" off the stock in 3 passes. The **X** value indicates that 1" diameter (0.500" radius) stock is in the machine. Each pass will be .0333"

The second command will also be 3 passes, but the first 2 will reduce the part by .04", and the last pass will be the remaining length of .02".

G83

Deep hole drilling **CANNED CYCLE**. The parameters include **X** position of the hole, **Z** final depth, and either **Q** (# of passes) or **K** (depth per pass).). After each stroke, **Z** retracts to the original Z-position.

Z should be 0.050" right of material before command.

EXAMPLE: G83 X0 Z-1 Q4

The above command drill a hole 1.000" deep in 4 equal strokes.

EXAMPLE: G83 X0 Z-1 K.3

The above command drill a hole 1.000" deep with (3) 0.300" strokes followed by one 0.100" stroke.

G87

Drill **CANNED CYCLE** with chip breaking. The parameters include **X** position of the hole, **Z** final depth, and either **Q** (# of passes) or **K** (depth per pass). After each stroke, **Z** retracts to the depth of the previous stroke only to break the chip.

Z should be 0.050" right of material before command.

EXAMPLE: G87 X0 Z-1 Q8

The above command will drill a hole 1.000" deep in 8 equal strokes.

EXAMPLE: G87 X0 Z-1 K.15

The above command drill a hole 1.000" deep with (6) 0.150" strokes followed by one 0.100" stroke.

G90

This code sets **Absolute Programming Mode**. In **Absolute Programming Mode** all positions are expressed as they relate to a *single zero reference point*. This code is modal and will remain in effect in the program until changed by a **G91** code. This is the default programming mode.

EXAMPLE: G90

G91

This code sets **Incremental Programming Mode**. In **Incremental Programming Mode** all positions are given in terms of relative distance and direction from the current tool position. This code is modal and will remain in effect until changed by a **G90** code. The default mode is **Absolute Programming Mode** (G90).

EXAMPLE: G91

G92

This code sets the initial starting point of the cutting tool. This is usually used at the beginning of a program when setting the **starting position**, however, it may be used at any **time to step** and **repeat a set of commands** at a new location.

EXAMPLE: G92 X0 Z-0.05

G95

This code chains to another part program. Include this command at the **end** of a part program file and it will automatically load and continue machining the specified program. Any code placed after this command will **not** be executed.

EXAMPLE: G95 #TEST

Will load and machine **TEST.CNC** out of the current directory at the end of the current file.

G96

This code sets a constant surface speed (CSS) of **xxx** ft/min.

EXAMPLE: G96 S80

will set a constant surface speed of 80 ft/min. The spindle speed will constantly adjust such that the surface cutting speed is 80 ft/min.

The speed parameter in the **G96 Snnn** command is in feet/minute. The speed parameter in the **M03 Snnn** command is in RPM. Given a radius of **x** inches, a constant surface velocity of **C** ft./min., and a rotational speed of **R** RPM:

$$C = (R \cdot \pi x) / 6 \text{ ft./min.}$$

and

$$R = (6 C) / (\pi x) \text{ RPM.}$$

Note: There may be an RPM limit on some machines. Once that RPM is achieved, the speed will not increase further.

Example: At a radius of 1 inch, for a constant surface velocity of 500 ft./min., the material should rotate at 954.93 RPM.

The following program demonstrates the constant surface velocity function.

```
G90          /ABSOLUTE COORDINATES
G00 X1      /INITIAL RADIUS = 1
M3 S954.93  /START SPINDLE
M00        /PAUSE
G96 S500    /START CONSTANT SURFACE VELOCITY OF 500 FT./MIN.
M00        /PAUSE. 500 FT/MIN = 954.93 RPM AT A RADIUS OF 1 INCH
G01 X2 F60  /THE RPM SHOULD DROP TO HALF THAT AT X=1
M0         /PAUSE
G01 X3 F60  /THE RPM SHOULD DROP TO 1/3 THAT AT X=1
M0         /PAUSE
G01 X4 F60  /THE RPM SHOULD DROP TO 1/4 THAT AT X=1
M0         /PAUSE
M02        /END PROGRAM
```

G97

This code will cancel constant surface speed (CSS) mode.

EXAMPLE: G97 S100

will cancel CSS mode and set the spindle to 100 RPM.

EXAMPLE: G97

will cancel CSS mode and the spindle will remain at its current speed.

Miscellaneous Functions (M-Codes):

M00

This command causes a **temporary stop**, by displaying a *Message Box*.

An optional message can be added to the M00 message box by using the comment character ("/"). Any text after the "/" will be displayed in the message box. Additionally, *user-defined variables*, or *system variables* may be displayed in the message by enclosing the variable name in parentheses "()".

Operation **resumes** by pressing the <OK> button.

Pressing <Cancel> will terminate the current program.

EXAMPLES:

```
M00
M00 / This message will be displayed in the message box.
M00 / Debug Only: Variable TestX = (TestX). TextZ = (TestZ)
M00 / Next Year will be [Year[0] + 1]
```

The first example displays a simple M00 message box. The second example includes the text after the "/". The third example displays the values of the user-defined variables **TestX** and **TestZ**. The fourth example displays an evaluated formula (next year), enclosed in square brackets.

M01

M01 is identical to **M00** with the exception that all *Coolant* and *Spindle* outputs (Outputs 1, 2, and 5), and any *VFD Spindle Controllers* (Teco 7300, or Fluxmaster 100) will be turned off while the message box is displayed, and restored to their original state upon completion of the command.

M02

This command causes an **end-of-program stop**. All operations are terminated and the system returns to the **main menu**.

M03

This turns <ON> the spindle motor **clockwise (control output #1)**

An optional S-parameter, for spindle speed, can be specified when using certain automatic spindles (Teco 7300 or Fluxmaster 100). Normally, **TurnMaster Pro** will wait for the spindle to reach the desired speed before continuing to the next command. If a negative S-parameter is encountered, **TurnMaster Pro** will not wait for the spindle to spool up before continuing.

EXAMPLES:

```
M03 / Turn Spinde ON, clockwise
M03 S100 / Turn spinde ON, clockwise (100 RPM). Wait for spindle to spool up.
M03 S-100 / Turn spinde ON, clockwise (100 RPM). Do not wait for spindle to spool up.
```

M04

This turns **<ON>** the spindle motor **counterclockwise (control output #5)**.

An optional S-parameter can be added, as described for the M03 command.

M05

This turns **<OFF>** the spindle motor (**control output #1 and #5**)

M06

This command selects the specified tool

EXAMPLE: M06 T2 / Selects Tool #2

Note: If you have specified a *Tool Turret* in **Options | Machine Parameters**, and you selected a tool number above the maximum tool stations of the turret, then **TurnMaster** will apply any X, and/or Z offsets for the new tool. The next time a tool, which is within the range of the tool turret, is selected, turret movement will commence from the last turret position.

M08

This turns **<ON>** **coolant (control output #2)**

M09

This turns **<OFF>** the **coolant (control output #2)**

M10

This turns **<ON>** the **vacuum (control output #3)**

M11

This turns **<OFF>** the **vacuum (control output #3)**

M12

This turns **<ON>** the **Auxiliary Output (control output #4)** This may be used to activate additional devices.

M13

This turns **<OFF>** the **Auxiliary Output (control output #4)**

M17

This command causes a return from subroutine. Must be used at the end of every subroutine. When executed this command will **restore** the **G90/91**, **G74/75** status and continue from the line following the **G25** subroutine call that invoked this subroutine.

M25

This command returns the tool to the Tool-Start (home) position.

If the current X-position is closer to the part than the X-start position, then **X axis** is moved first, otherwise the **Z-axis** is moved first.

M29

This command homes to the limit of one axis (X or Z axis) and returns the result.

EXAMPLE: M29 x (varname)

If in absolute programming mode then the x or z position is returned in *varname*. Otherwise the incremental movement (the distance from the last position to the limit) is returned in *varname*.

M39

This **<CLOSES>** the chuck (**control output #6**)

M40

This **<OPENS>** the chuck (**control output #6**)

M41

This turns **<ON>** the Auxiliary Output #3 (**control output #7**). This may be used to activate additional devices.

M42

This turns **<OFF>** the Auxiliary Output #3 (**control output #7**).

M43

This turns **<ON>** the Auxiliary Output #4 (**control output #8**). This may be used to activate additional devices.

M44

This turns **<OFF>** the Auxiliary Output #4 (**control output #8**).

M66

This allows for jogging during g-code execution without affecting the counters.

EXAMPLE: M66

Note: When **TurnMaster** encounters an **M66** command in G-code, the Jog Tool window will appear on screen. When you jog, the counters will change. As soon as you close the Jog Tool window, the counters will be restored to the exact values they were before the **M66** command was encountered, but no motion will occur.

M94

This command sends a *literal* command to the **MN400** Controller. The command should be a native **MN400** command. **TurnMaster** does not validate or parse the string before passing it to the **MN400**. The command may be embedded in double quotes but this is not required.

Note: The **M94** command is ignored if any other controller is selected, or if not in *Machine* mode.

EXAMPLES: M94 m400,0,-200 / Move X 400 steps, Y -200 steps
 M94 "m400,0,-200"
 M94 c2,0 / Turn Output #2 ON

M97

This command pauses **part program execution** until the specified input is active (low voltage). Valid input #s are I1 - I8.

Note: This feature is only available with the **OptiStep Plus** and **MN400** controllers.

EXAMPLE: M97 I2

Will wait for **input 2** to become active (low voltage) before continuing program execution.

M99

This command restarts execution of the **part program** from the **beginning**.

Function Quick Reference

Preparatory Functions (G-Codes)

G00	Rapid positioning move
G01	Linear cutting move
G02	Clockwise circular cutting move
G03	Counterclockwise circular cutting move
G04	Set dwell in seconds
G25	Execute subroutine
G26	Conditional branch
G27	Unconditional branch
G28	Set system or user defined variable to value
G33	Canned threading cycle
G60	Turn off Continuous Contouring mode
G64	Turn on Continuous Contouring mode
G70	Set inch programming (default)
G71	Set metric programming
G74	Sets single quadrant arc programming mode
G75	Sets multiple quadrant arc programming mode (default)
G80	Cancel canned cycle
G81	Canned cycle for turning down outside diameter
G82	Canned cycle for reducing part length (facing)
G83	Canned cycle for deep hole drilling
G87	Canned cycle for deep hole drilling with chip breaking
G90	Set absolute programming mode (default)
G91	Set incremental programming mode
G92	Set current tool position counters to value
G95	Chain to next part program
G96	Set Constant Surface Speed (CSS) mode
G97	Cancel Constant Surface Speed (CSS) mode

Miscellaneous Functions (M-Codes)

M00	Temporary stop
M01	Temporary stop - Coolant & Spindle OFF
M02	End of program stop
M03	Spindle <ON> CW (output #1)
M04	Spindle <ON> CCW (output #5)
M05	Spindle <OFF> (output #1, output #5)
M06	Tool change
M08	Coolant <ON> (output #2)
M09	Coolant <OFF> (output #2)
M10	Vacuum <ON> (output #3)
M11	Vacuum <OFF> (output #3)
M12	Auxiliary Output <ON> (output #4)
M13	Auxiliary Output <OFF> (output #4)
M17	Return from subroutine
M25	Rapid traverse to home position
M29	Home to Axis Limit
M39	Chuck <CLOSE> (output #6)
M40	Chuck <OPEN> (output #6)
M41	Auxiliary Output #3 <ON> (output #7)
M42	Auxiliary Output #3 <OFF> (output #7)
M43	Auxiliary Output #4 <ON> (output #8)
M44	Auxiliary Output #4 <OFF> (output #8)
M66	Enables Jogging during a program
M94	Send literal command to MN400 Controller
M97	Wait for true input state then continue
M99	Restart part program from beginning

Configuration and Setup

This section will instruct you on how to configure **TurnMaster Pro** to work with your turning machine. This section will only cover the procedures that are **necessary** to successfully machine a part. Enter the values as indicated below.

Entering Part Program Information

It is necessary to enter information about your part program into **TurnMaster Pro** so that it can accurately simulate the making of your part. Enter the values shown if you want to run the tutorial in Section 5.

For Metal Tutorial:

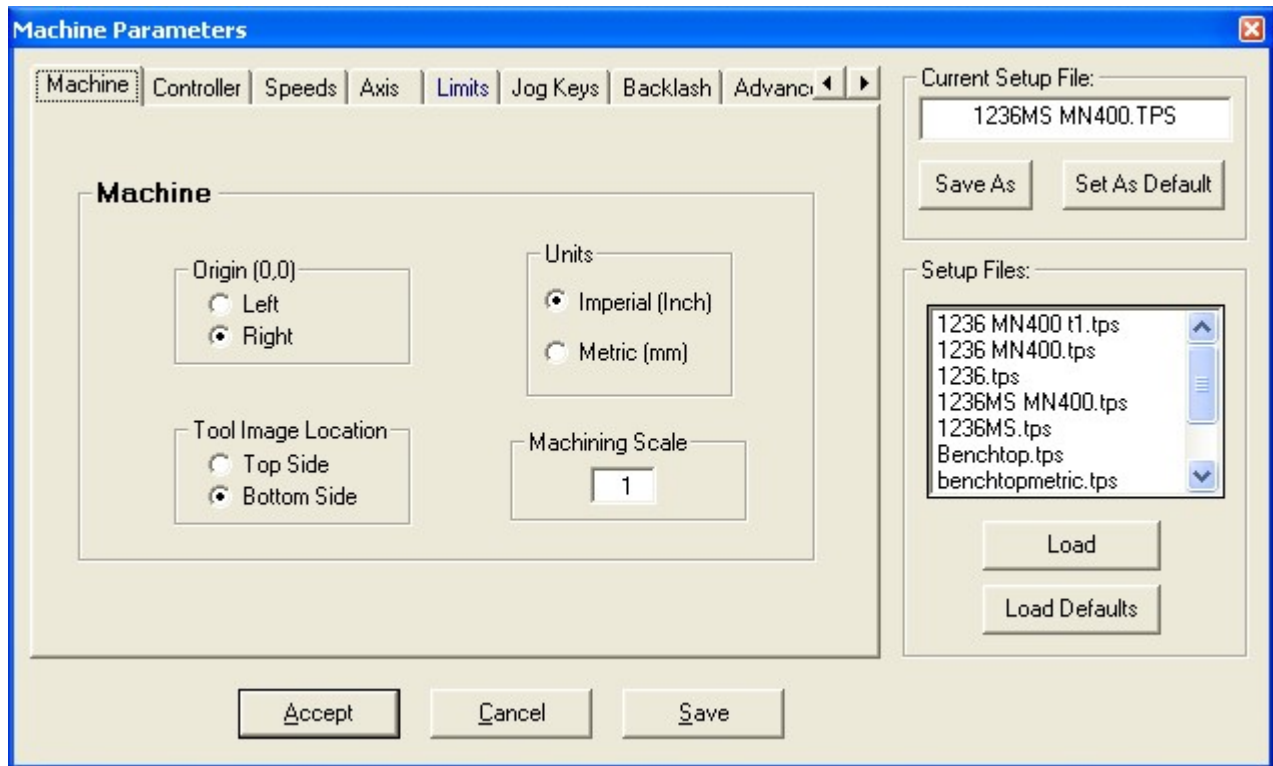
1. Select the type of **Units (inch)**, the **Fixturing Method (chuck)**, and the **Location of the Origin** to the **RIGHT**.
2. Enter the **Material Length** in the specified **Units** Enter 2.0000.
3. Enter the **Stock Outside Diameter** in the specified **Units**. Enter 1.0000.
4. Enter the **Stock Inside Diameter** in the specified units. Enter 0.5.
5. Enter the **Machining Scale**. Entering **1** means "TO SCALE" (See **Option Menu Section for Further Information**). Enter 1.
6. Choose **[Accept]**.

For Wood Tutorial:

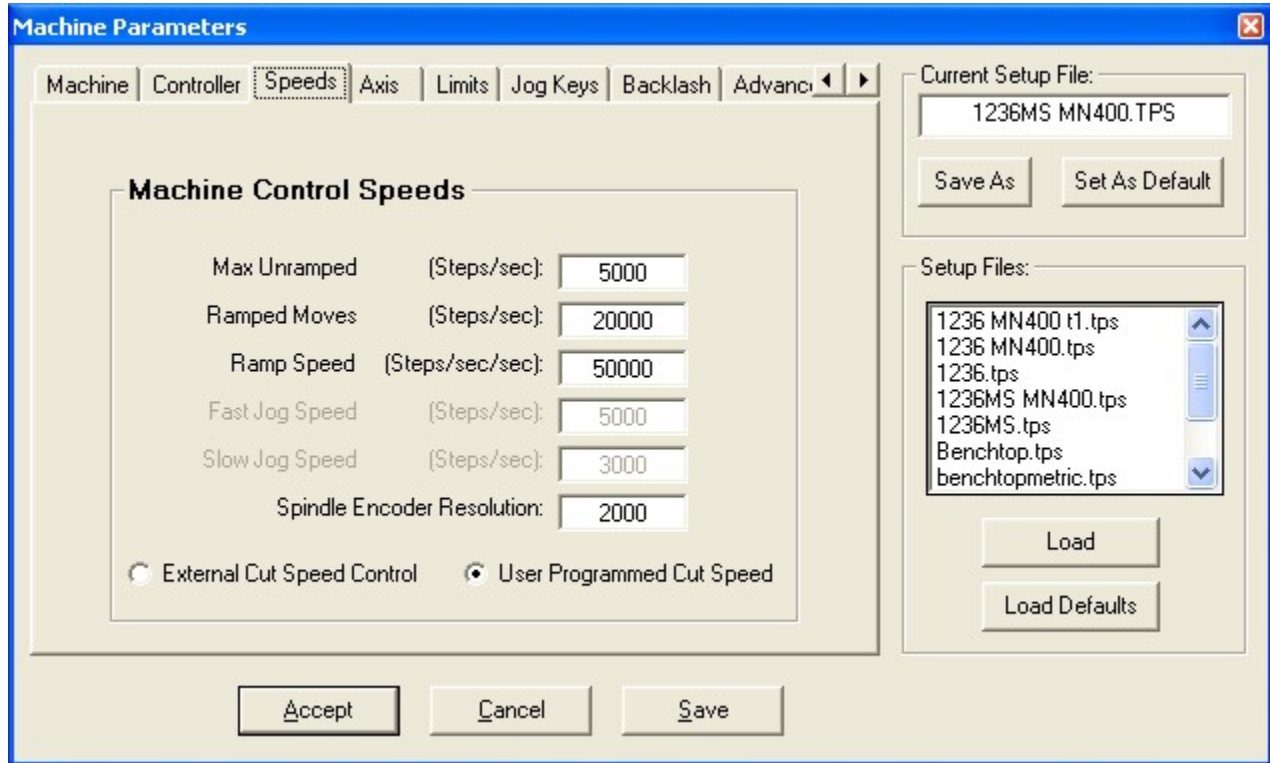
1. Select the type of **Units (inch)**, the **Fixturing Method (centers)**, and the **Location of the Origin** to the **LEFT**.
2. Enter the **Material Length** in the specified **Units** Enter 5.0000.
3. Enter the **Stock Outside Diameter** in the specified **Units**. Enter 1.1250.
4. Enter the **Stock Inside Diameter** in the specified units. Enter 0.
5. Enter the **Machining Scale**. Entering **1** means "TO SCALE" (See **Option Menu Section for Further Information**). Enter 1.
6. Choose **[Accept]**.

Setting the Machine Control Speed

The following graphic shows the **Machine Parameters Window** in **TurnMaster Pro**. This is where **ALL** machine and program settings should be made and/or adjusted.



The **TurnMaster Pro** software has to know the speeds your machine can move at **Reliably**. Choose the **Options-Machine Parameters** Menu with the mouse or type **<ALT>-O M**. Click on the **Speeds** tab.

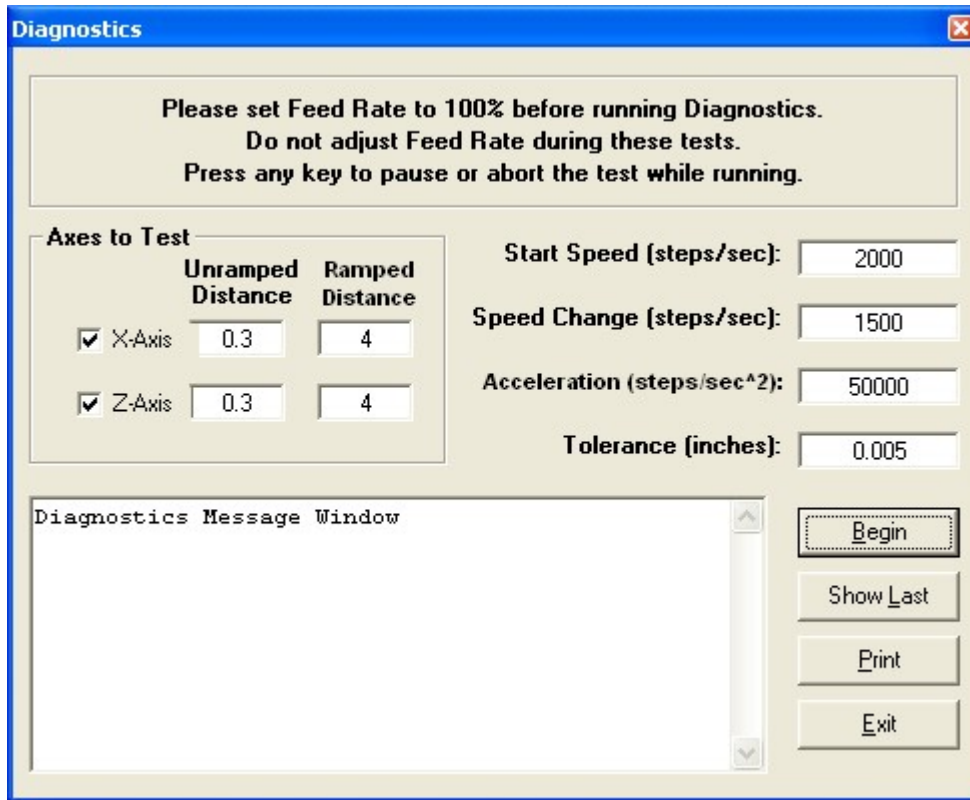


1. Set the **Max Unramped Speed** to the maximum speed that the motors can *instantaneously* accelerate to. This speed is in steps per sec and can easily be converted from inches per minute by using the following formula:

$$\text{Steps/Sec} = \frac{\text{Screw Pitch} \times \text{\#Steps/Rev}}{60} \times \frac{\text{Inches}}{\text{Minute}}$$

2. Set the **Max Ramped Speed** to the maximum speed at which your machine can move **reliably**. The default is set to **3,500 Steps/Sec**. You may be able to set this at a much higher rate if you are using ballscrews or other low friction mechanical drives.
3. The **Ramp Speed** is an acceleration parameter and is usually set for **5,000 Steps/sec/sec** which is a good all-purpose value.
4. Set the **Fast Jog Speed** to the same speed as the **Max Ramped Speed**.
5. Set the **Slow Jog Speed** to a speed that can be used for fine positioning. A good value to start with is 300. Adjust the value as needed.

** The **Control-Diagnostics** option can be used to automatically find the **maximum unramped** and **ramped speeds** of your machine. Press **<ALT>-C D** on the keyboard, select the axes to be tested, and enter the test parameters. Choose **[Accept]** to start the test. The following graphic shows the **Diagnostics Window**.



Axis Control Parameters

Choose **Options-Machine Parameters**, with the mouse or type **<ALT>-O M**. Click the **Axis** tab

1. Set the **Lead Screw Pitch** for both axes to the proper value for your machine. The default is 20 threads per inch (TPI) for our Sherline sized machines, and 5 for our Lathe 1236 and Lathe 1340. Some typical values for ballscrews and ACME threads are 5 and 10 TPI. Enter this value according to your screw.
2. Set the **Logical Steps Per Revolution** for both axes to the proper value for your machine. The default value is **400 Steps/Rev**. You can calculate this value if you know the number of degrees per step of the motor:

$$\text{Steps/Rev} = \frac{720(\text{or } 360 \text{ if fullstepping})}{\text{\# of degrees per step}}$$

3. Set the **Polarity** of each axis. A **NORMAL Polarity** is defined as: if the axis turns counterclockwise as viewed from the motor end, a positive cutting move will result. Set the axis to **NORMAL** if the above rule is true and **REVERSE** if it is false. **Note:** If **REVERSE** is selected for any axis, then the limit switch connections must also be reversed (i.e. connect a positive limit switch to the Negative Input).

Setting the Limit Sensors

Choose **Options-Machine parameters-Limit Sensors** with the mouse or type **<ALT>-O L**.

****NOTE: This is to be done only if the machine you are using has limit sensors.**

Choose the **Limit Sensor** (either + or -) for each axis that is to be considered home. These will be the two sensors that the machine goes to in homing moves.

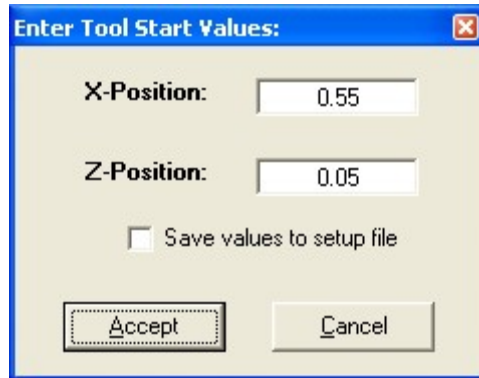
4.5 Setting the Tool Offsets

NOTE: Skip this section if your machine has only one cutting tool. This procedure will step you through process to determine the tool offsets. *Verify that the machine is connected properly and that the power is "ON" before continuing!*

1. Install the right hand cutting tool (Tool #1) in the machine.
2. Chuck up a 0.5" diameter by 2" long stock.(Brass or aluminum would be ideal)
3. Jog the tool to the right edge of the part and face it off.
4. Exit jog mode and select Control - set new position.
5. Enter a value of zero for the Z axis only and accept that.
6. Jog the tool back out to near the diameter surface and take a shallow cut (about 0.010 to 0.015 will do).
7. Measure the diameter with a micrometer or calipers and divide by 2.
8. Exit Jog Mode and select Control - set new position. Enter this radius value in the X axis field and accept it
9. Select Tools-Toll Start Position and enter it as 1/2 the diameter + .050" for the X and 0.050" for the Z axis.. Choose the save checkmark and accept it.
10. Select Control - Return Tool to Start.
11. If you have limit switches installed on your machine, you can select Control- Reprogram Limit sensors to have it remember the start position as a reference.
12. The offset for tool #1 is always zero. To program the offsets for other tools, switch out to each subsequent tool and jog it to the surface for touch off and observe the counter at this position. Subtract the two numbers and place the value in the tool offset for that tool.
13. Repeat for additional tools, using the #1 tool as your point of reference for each one.

Setting the Start Position

1. Choose **Tools-Tool Start Position**. Enter the absolute coordinates of where the tool will start from at the beginning of the machining process. This value should match the G92 code in the CNC program file that is to be run. Typically, it is away from the stock in both directions common values used are 0.050 or 0.100".



For the Metal Tutorial

Set it for: X-axis = 0.550"
 Z-axis = 0.050"
 to work with the tutorial.

For the Wood Tutorial

Set it for: X-axis = 0.750"
 Z-axis = 0"
 to work with the tutorial.

Choose **[OK]** when complete. These values may be changed to suit your specific needs.

2. **Save parameters as default.** This will store all your specific parameters on disk so that they automatically load at start-up.

Congratulations, the software is now configured to your specific machine. In the future you will only need to redo this if you change your tool setup.

Tutorial

(Metal Turning)

This section will show you how to design and write a **part program** as well as how to graphically simulate and manufacture the **part** using **TurnMaster Pro**. This **tutorial** will touch on basic machining functions such as **linear** and **circular cutting moves**, **rapid traverse moves**, and **homing moves**.

Before attempting to create and manufacture this **part**, your machine should be **setup** properly (see **Section 4 - Configuration and Setup**). A drawing of the **part** that will be produced is shown in **Figure 5.1.1**.

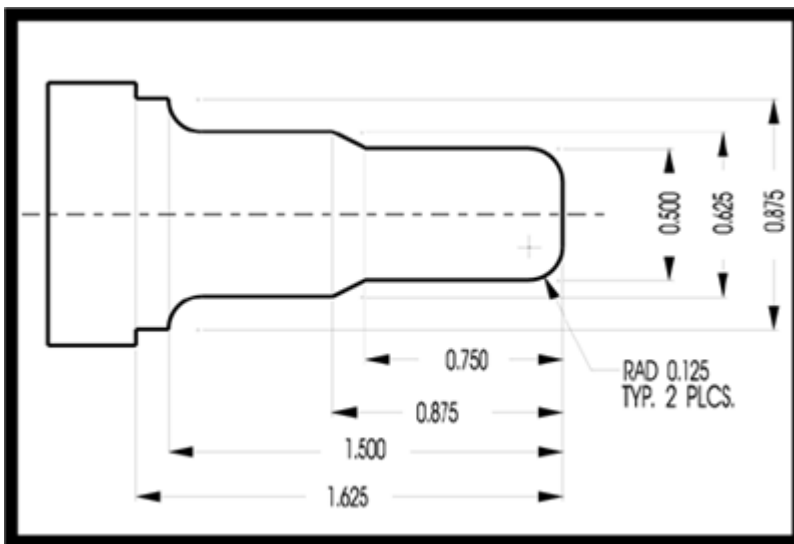


Figure 5.1.1 Part Profile Drawing

5.1 Writing a Metal Part Program

Start **TurnMaster Pro** and click **View** and then select **CNC Edit Mode**. You can easily **create**, **edit**, and **save** your **part** designs from this built in editor.

1. Enter the first three lines. They set the **programming mode**, **starting position** of the machine, and turn on the **spindle motor**. These three commands are usually included in most **part programs**. The comments are optional.

```
G90                                /SET PROGRAMMING MODE
G92 X.550 Z.050                   /SET STARTING POSITION
M03                                /TURN ON SPINDLE MOTOR
```

2. Enter the next three lines. They face the stock, set the **feedrate**, and return the machine to the **starting position**.

```
G00                                /FACE THE STOCK
G01 X-.005 F30
M25                                /RETURN TO THE STARTING POSITION
```

3. Enter the next 38 lines. They rough out the material to within 0.005" of the finished **part profile**.

```
G00 X .470           /BEGINNING OF ROUGHING CUTS
G01                Z -1.620
G01 X .520
G00                Z .050
G00 X .440
G01                Z -1.620
G01 X .500
G00                Z .050
G00 X .410
G01                Z -1.492
G01 X .450
G00                Z .050
G00 X .380
G01                Z -1.481
G01 X .420
G00                Z .050
G00 X .350
G01                Z -1.459
G01 X .390
G00                Z .050
G00 X .320
G01                Z -1.409
G01 X .360
G00                Z .050
G00 X .285
G01                Z -.815
G01 X .330
G00                Z .050
G00 X .255
G01                Z -.755
G01 X .340
G00                Z .050
G00 X .255
G01                Z -.045
G01 X .350
G00                Z .050
G00 X .195
G01                Z -.016
M25                /RETURN TO STARTING POSITION
```

The **roughing cuts** were all calculated using **basic math** and **right angle trigonometry**.

4. The remaining lines are the **finishing cuts**.

```
G00 X .050 Z .010           /BEGINNING OF FINISH CUTS
G01 X 0 Z 0
G01 X .125                 /MACHINE TO FIRST RADIUS
G03 X .250 Z -.125 K .125  /MACHINE FIRST RADIUS
G01 X .250 Z -.125         /MACHINE TO THE TAPER
G01 X .3125 Z -.875       /MACHINE TAPER
G01 X .3125 Z -1.375      /MACHINE TO SECOND RADIUS
G02 X .4375 Z -1.500 I .125 /MACHINE SECOND RADIUS
G01 X .4375 Z -1.625
G01 X .510
M25                        /HOME TOOL
M02                        /END OF PROGRAM
```

5. Choose **File-Save**. Type in **TUTORMTL** and click **Save**. This will **save** the design in the current directory as **TUTORMTL.CNC**.

Some other helpful hints:

1. **Optional N-Sequence Numbers**. Each program line may be numbered with a **Nxxxx** sequence code. **xxxx** can be any number from **0001** to **9999**.
2. **Align your code in columns**. This will make it easier for you to examine other portions of your program and to refer to it in the future.
3. **Add frequent comments**. Especially where the command itself does not explain the operation you are performing.

5.2 Machining a Metal Part Program

Now that you have entered the **part program**, you will be instructed in this section on how to actually produce it. Before attempting to **manufacture** this **part**, your machine should be **setup** properly (see **Section 4 - Configuration and Setup**).

The following equipment is needed for this exercise:

- A **lathe** with a **3-jaw chuck**.
- A standard **right hand cutting tool**.
- 1" Dia. stock (**brass, plastic, or mild steel**).

Start **TurnMaster Pro 2014**. The screen shown in **Figure 5.2.1** should be on your desktop.

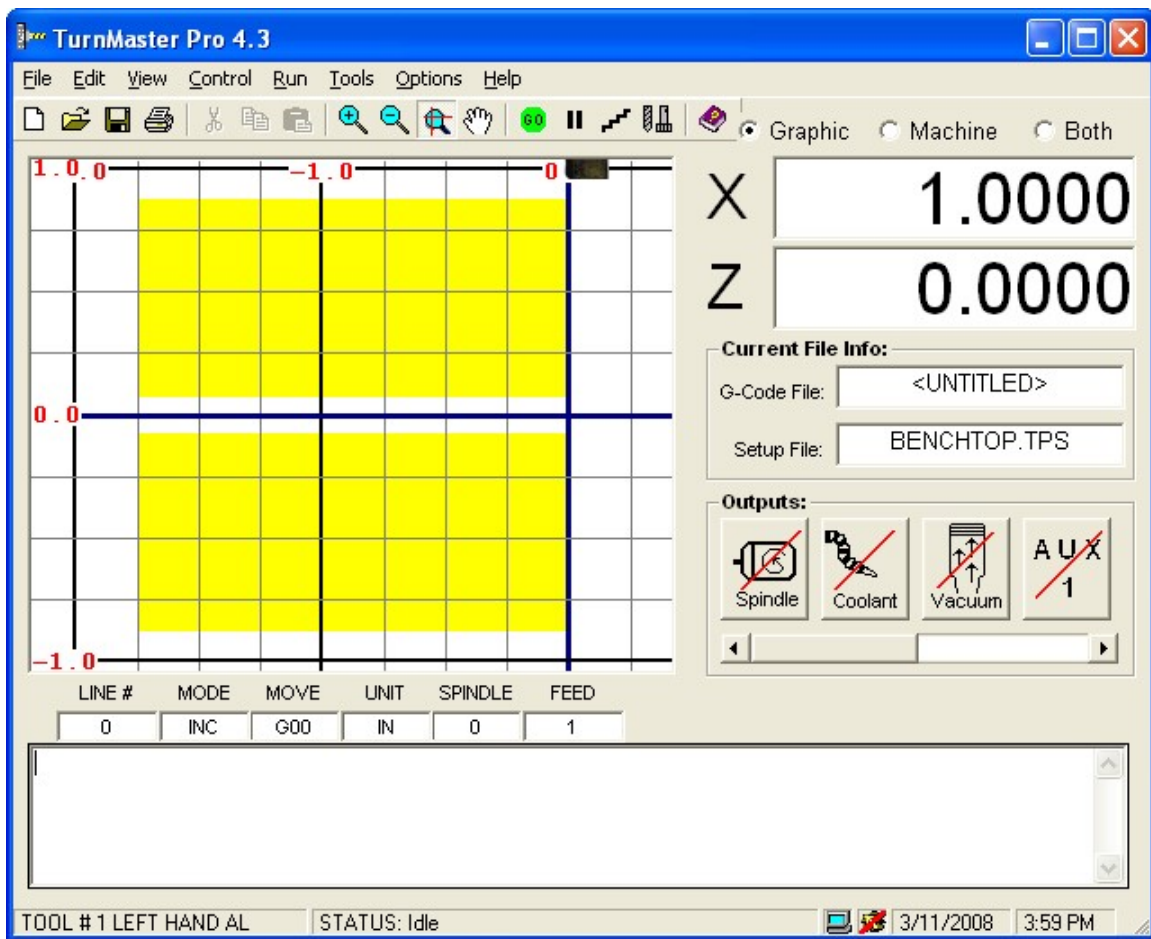


Figure 5.2.1 TurnMaster Pro Graphical Screen

1. Choose **Options-Material Size/Setup**. Check that the **options** and **values** are set as follows:

Material Length = 2.000"
 Outside Diameter = 1.000"
 Inside Diameter = 0.000"

Choose Options-Machine Parameters. Check that the options and values below are set as follows:

Units = Inch
 Origin 0,0 = Right
 Machining Scale = 1

Save these parameters as **default**.

2. Choose **File-Load** with the mouse or **<ALT>-F L** from the keyboard. Select **TUTORMTL.CNC** and choose **[OK]**. The program file is now loaded into **TurnMaster Pro**.

- Choose **Run-Auto Run** with the mouse or **<ALT>-R A** from the keyboard. This runs the **turning simulation program**. You should see the **tool** cutting the stock as the program lines scroll at the bottom of the screen. Press the **<spacebar>** after a few lines have executed to pause the simulation. In the **dialog box** choose the **[STEP]** option. You are now in **STEP** mode, click **[STEP]** again when prompted and the next program line will be executed.

This is a useful **tool** in **troubleshooting** a design problem. Hit the **<spacebar>** again and choose **[CONTINUE]**. The **part program** should be free running again. Once it has completed running, it should look like **Figure 5.2.4**.

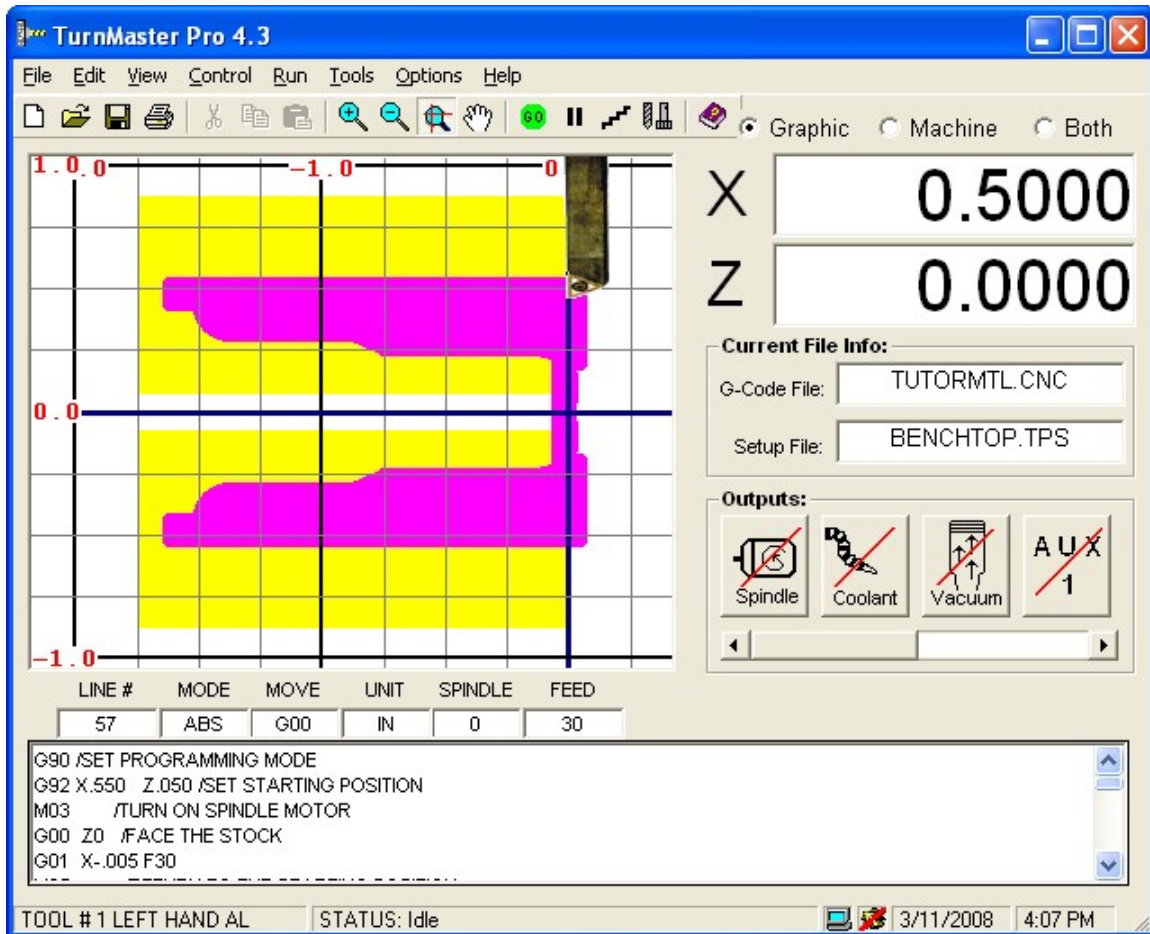


Figure 5.2.4 Tutorial CNC Simulation

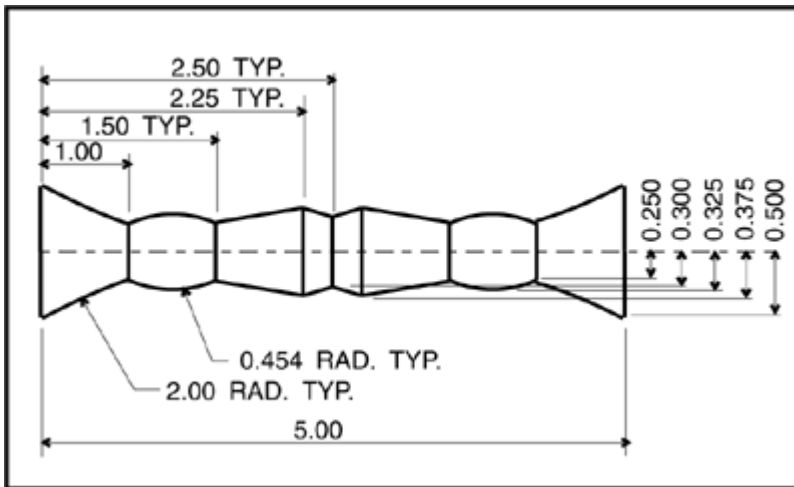
If your screen does not look like **Figure 5.2.4**, use **CNC Edit mode** to check and **edit** the **CNC part program**.

- Click **Machine Mode** with the mouse. Note that the status line at the bottom of the screen indicates **machine mode** as well as the radio button selection in the top right corner of the window.
- If you have **limit sensors** on your machine, choose **Control-Reposition From Limits**. This will automatically move your **tool** to the **start position**. If you do **NOT** have **limit sensors**, **jog** your **tool** to the **start position**.
- Choose **Run-Auto Run** again. **TurnMaster Pro** will now produce the **part**.

NOTE: The **<spacebar>** will pause the machining process at any time. If something starts going wrong during machining, **PRESS THE EMERGENCY STOP** button on your DriveRack!

(Wood Turning)

5.3 Writing a Wood Part Program



1. Enter the first three lines. They set the **programming mode**, **starting position** of the machine, and the **arc mode**. These three commands are usually included in most **part programs**. The comments are optional.

```
G90                                /SET THE PROGRAMMING MODE
G92  X .750  Z .050                /SET STARTING POSITION
G75                                /SET MULTIPLE QUAD ARC MODE
```

2. Enter the next 2 lines. They turn on the **spindle motor** and set the **tool number**.

```
M03
M06  T3
```

3. Enter the next 24 lines. They rough out the material to within **0.100"** of the finished **part profile**. Notice the **feedrate** is being set in the third line.

```
G00  X .6280
G00                                Z .1533
G01  X .5280                        F60
G01                                Z 4.8467
G01  X .6310
G00                                Z 4.6568
G01  X .4280
G01                                Z .3432
G01  X .5310
G00                                Z .6089
G01  X .3280
G01                                Z 1.9480
G01  X .4310
G00  X .4280
G00                                Z 2.4187
G01  X .3280
G01                                Z 2.5813
G01  X .4310
G00  X .4280
```



```

G00          Z 3.0520
G01 X .3280
G01          Z 4.3911
G01 X .4310
M25          /RETURN TO STARTING POSITION

```

The **roughing cuts** were all calculated using **basic math** and **right angle trigonometry**.

4. The remaining lines are the finishing cuts.

```

G00 X .6280
G00          Z .0000
G00 X .6250
G03 X .2500  Z 1.0000  I 1.6172  K 1.1768
G02          Z 1.5000  I -.3792  K .2500
G01 X .3750  Z 2.2500
G01 X .3000  Z 2.5000
G01 X .3750  Z 2.7500
G01 X .2500  Z 3.5000
G02          Z 4.0000  I -.3792  K .2500
G03 X .6250  Z 5.0000  I 1.9922  K -.1768
G00 X .6280
G00 X .7500
G00          Z .0000
M05
M25
G74
M02

```

5. Choose **File-Save**. Type in **TUTORWOD** and press **<ENTER>**. This will **save** the design in the current directory as **TUTORWOD.CNC**.

The differences between the **metal turning** example and the **wood turning** example are as follows:

1. The material size in the metal example is 2" long and 1" in diameter, and the wood example is 5" long and 1.25" in diameter.
2. The selected **tool** in the metal example is **Tool #1 (right hand cutting tool)** and in the wood example it is **Tool #3 (universal cutting tool)**.
3. The **starting position** in the metal example is X0.550" and in the wood example it is X0.750 and Z0.

5.4 Machining a Wood Part Program

Now that you have entered the **part program**, you will be instructed in this section on how to actually produce it. Before attempting to manufacture this **part**, your machine should be **setup** properly (see **Section 4 - Configuration and Setup**).

The following equipment is needed for this exercise:

- A **lathe** equipped to turn between **centers**.
- A standard **universal cutting tool**.
- 1.25" Dia. stock (wood).

1. Choose **Options-Material Size/Setup**. Check that the **options** and **values** are set as follows:

Material Length = 5.000"
Outside Diameter = 1.250"
Inside Diameter = 0.0000"
Units = Inch

Choose **Options-Machine Parameters**. Check that the **options** and **values** are set as follows:

Origin 0,0 = Left
Machining Scale = 1

Save these parameters as **default**.

2. Choose **File-Load** with the mouse or **<ALT>-F L** from the keyboard. Select **TUTORWOD.CNC** and choose **[OPEN]**. The program file is now loaded into **TurnMaster Pro**.
3. Choose **Run-Auto Run** with the mouse or **<ALT>-R A** from the keyboard. This runs the **turning simulation** program. You should see the **tool** cutting the stock as the program lines scroll at the bottom of the screen. Press the **<spacebar>** after a few lines have executed to pause the simulation. In the **dialog box** click the **[STEP]** option. You are now in the **STEP** mode. Click **STEP** again when prompted and the next program line will be executed.

This is a useful **tool** in **troubleshooting** a design problem. Hit the **<spacebar>** again and choose **[CONTINUE]**. The **part** program should be free running again. Once it has completed running, it should look like **Figure 5.4.4**.

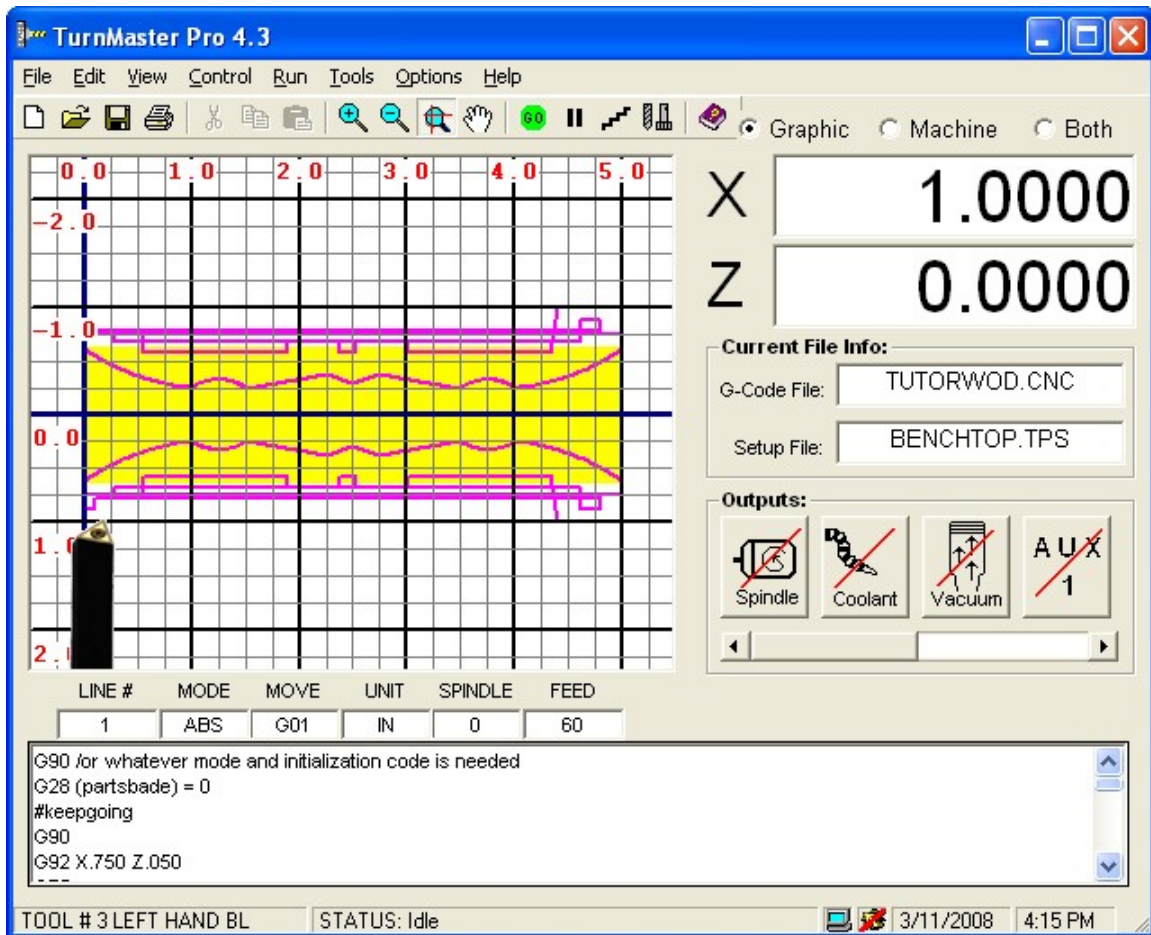


Figure 5.4.4 Tutorial CNC Simulation

If your screen does not look like **Figure 5.4.4**, use the **CNC Editor** to check and **edit** the **CNC part program**.

4. Click **Machine Mode** with the mouse. Note that the status line at the bottom of the screen indicates **machine mode** as well as the radio button selection in the top right corner of the window.
5. If you have **limit sensors** on your machine, choose **Control-Reposition From Limits**. This will automatically move your **tool** to the **start position**. If you do **NOT** have **limit sensors**, **jog** your **tool** to the **start position**.
6. Choose **Run-Auto Run** again. **TurnMaster Pro** will now produce the **part**.

**** NOTE:** The **<spacebar>** or the **<Esc>** key will pause the machining process at any time.

If something starts to go wrong with the machining, use the **<spacebar>** to pause or **PRESS THE EMERGENCY STOP** button on your DriveRack or other controller!

Using the Math Formula Evaluation Feature

Beginning with **Millmaster Pro** and **TurnMaster Pro** Versions 4.1.xxx, a math formula evaluation feature was added to the G-Code parser. This math evaluation feature combined with user-defined variable names is sometimes referred to as Parametric Programming.

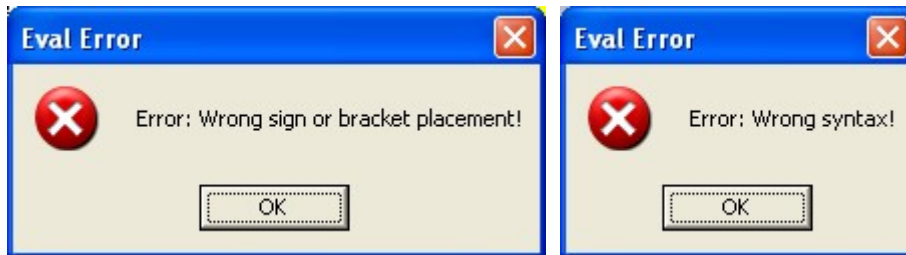
Parser:

Internally, the G-Code parser first converts any system variables or user-defined variables, in the G-Code line being parsed, to their equivalent numerical values.

The parser then looks for any square bracket pairs [...]. If found, the parser will attempt to evaluate the formula inside the brackets.

If the formula is a legal math expression, it will be converted internally to the equivalent numerical value and executed. For example, the G-Code line `G00 X[1+2]` would be executed as `G00 X3`.

If the formula is not valid, an error message is displayed, and the parser returns a value of **0** for the expression. For example:



Note: If the formula is not valid, the G-Code program continues to execute with a zero value returned from the expression. Be sure to test all G-Code programs in graphics only mode, and syntax checking turned on before running any programs in machine mode.

Syntax:

Multiple bracket pairs can exist on a single line. They may exist separated by parameters `G01 X[2*3] Y[1-.2]` or they may be nested to any level within the same formula `G00 X[1-[0.1*2]]`.

You must, however, make sure the entire expression is within the brackets. For example, `G00 X 1+[1-0.1]`, or `G00 X[1-0.5]+1` would both be invalid. This type of error would not be caught by the evaluation parser since the data inside the bracket pair is valid. In the examples above the program would continue to run, but with the wrong values for X.

Also, do not use the parenthesis characters () inside the expression. Parenthesis are reserved for system variables or user-defined variables. For example:

```
G28 (X1) = 0.1 /Set User Defined Variable X1 = 0.1
G92 X(X1) Y[1-(X1)] Z[0+[0+0.050]] /ESTABLISH START POINT
```

would establish the start point at X0.1, Y0.9 and Z0.5.

However the line:

```
G92 X(X1) Y[1-(0+0.1)] Z[0+[0+ .050]] /ESTABLISH START POINT
```

would be invalid since the value inside the parenthesis (**0+0.1**) is not a valid variable name.

Functions:

The following arithmetic functions are valid:

Arithmetic Operators:

+	Addition
-	Subtraction
*	Multiplication
\	Division (do not use the / char as it is reserved for comments)

abs[x]	Absolute Value of x
fix[x]	Integer portion of x (negative numbers round up)
int[x]	Integer portion of x (negative numbers round down)
rnd[x]	Random number between 0 and 1. X is the random # seed.
sgn[x]	Sign of x. Returns 1 if positive, -1 if negative, 0 if zero.

arcsin[]	Arcsine (in radians)
arccos[]	Arccosine
arcsec[]	Arcsecant
arccosec[]	Arccosecant
arccotan[]	Arccotangent
atn[]	Arctangent
sin[]	Sine
cos[]	Cosine
cosec[]	Cosecant
cotan[]	Cotangent
sec[]	Secant
tan[]	Tangent

exp[x]	Inverse Natural Logarithm or the base of a natural logarithm.
log[x]	Natural Logarithm of x
sqr[x]	Square Root of x

timer[0]	Seconds since midnight (dummy parameter [0] is required)
Year[0]	Current year
Month[0]	Current month
Day[0]	Current day

Logical Operators:

=	Equal To
>	Greater Than
<	Less Than
>=	Greater Than or Equal To
<=	Less Than or Equal To

Note: The logical operators are best used with the G26 (Conditional Jump) command. If True, the jump occurs, otherwise execution falls to the next line in the G-Code program.

However, these logical operators will return an arithmetic -1 if the condition is True, and 0 if the condition is false.

Also, be sure the entire logical expression is surrounded by square brackets.

For example:

```
[1 <= 2] would return -1 (True)
[99 = [33*3]] would return -1 (True)
[5 > [3*2]] would return 0 (False)
```

Examples:

<u>G-Code Line</u>	<u>Evaluates To</u>
G00 X[1+0.1]	G00 X1.1
G01 X[2*.1] Y[1\4]	G01 X0.2 Y0.25
G00 X[1+[1.1*2]]	G00 3.2
G92 X[cos[3.1416]] Y[sqr[4]]	G92 X-1 Y2
G28 (CurrentYear)=[YEAR[0]]	G28 (CurrentYear) = 2008
G28 (NextYear)=[(CurrentYear)+1]	G28 (NextYear)=2009
G01 [(NextYear)-(CurrentYear)]	G01 1
G28 (blnTest) = [(CurrX) >= 1]	Sets var (blnTest) to -1 (True) if (CurrX) >= 1, else (blnTest) is set to 0 (False).
G26 [(NumPasses) <= 0] #Done	Jump to #Done if (NumPasses) <= 0.

Sample Program:

```
/-- Perform external taper with Unidirectional Roughing.
/ The startup variables (StartX), (StartY),
/ (EndX), (EndY), and (DepthPerCut) determine the
/ dimensions of the taper.
/
/ To change the taper dimensions, simply change the
/ initial value of these variables at the start of
/ the program.
M03
G90 /Absolute Mode

/-- Taper specific vars. Change these vars to change taper.
G28 (StartX)=1 /-- Start Point of taper
G28 (StartZ)=0
G28 (EndX)=0.5 /-- Taper Diameter = 1 - 0.5
G28 (EndZ)=-2 /-- Taper Width = 0 - (-2)
G28 (DepthPerCut)=0.1

/-- Working variables. Do not change these vars.
G28 (NumPasses) = 1 /-- First Pass

/-- Get the direction the taper is moving. (Either +1, or -1).
G28 (Direction) = [sgn[(StartX) - (EndX)]]

/-- Compute the ratio of Delta Z, and Delta X
G28 (XZRatio)=[[(StartZ)-(EndZ)] \ [(StartX)-(EndX)]]

G0 X(StartX) Z(StartZ) / Move to initial cutting position
```

```

/-- Main Loop. Each iteration will cut the next level.
#NextCut

/-- Compute the depth of the next pass.
G28 (NextX)=[(StartX) - [[(DepthPerCut) * (NumPasses)] * (Direction)]]

/-- If this next computed cut is past the Ending X value,
/ then we're finished. We have to take the direction into
/ account since X values may be getting larger or smaller.
G26 [[[ (EndX)-(NextX)]*(Direction) ] > 0] #FinishedTaper

/-- Next X is good, so continue.
G1 X(NextX)

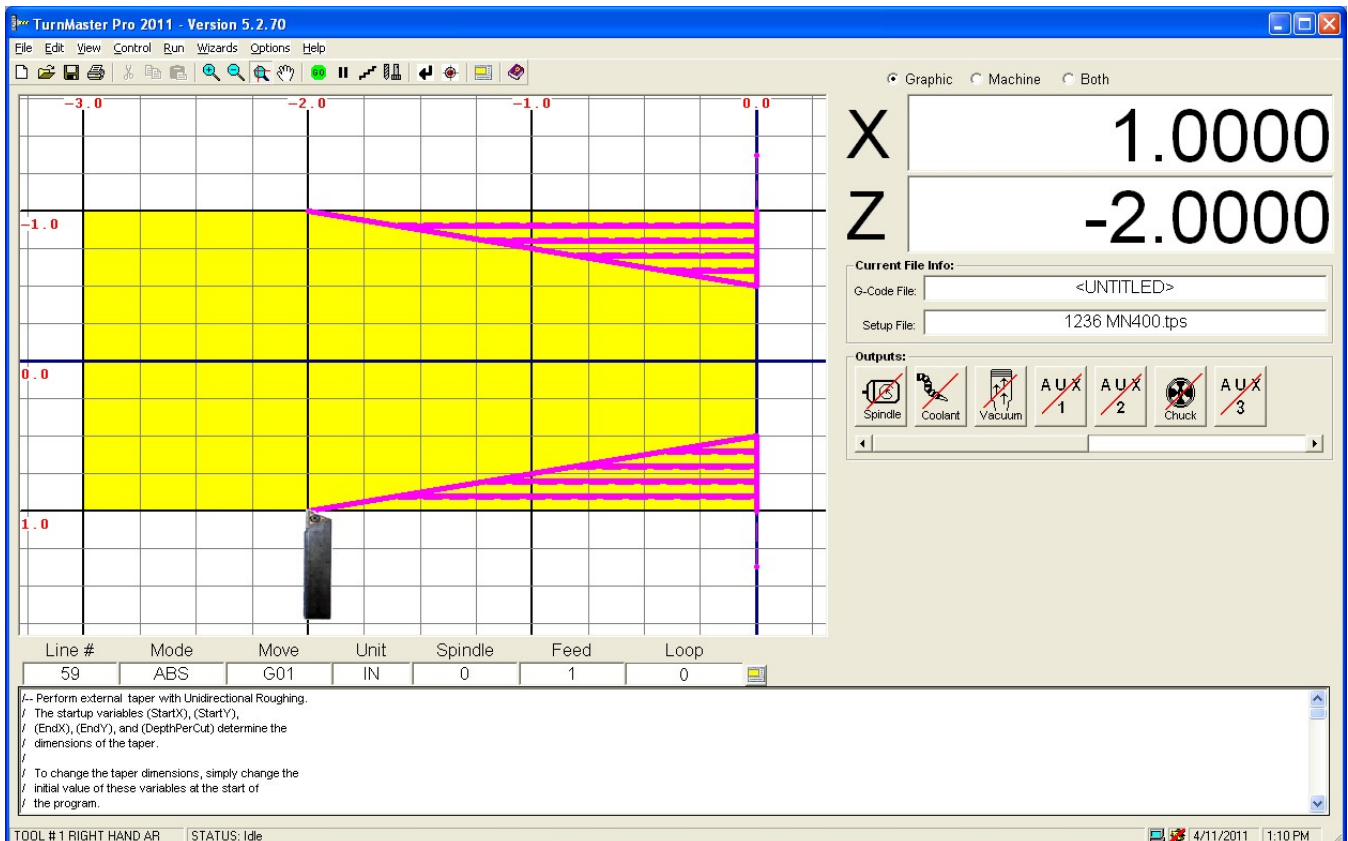
/-- Compute ending Z position for this value of X
G28 (TempZ)=[(EndZ) - [[(NextX)-(StartX)]*(XZRatio)]]
G1 Z(TempZ)

/-- Now back out and move back to Starting Z.
G0 X[(NextX)+0.02] /-- Back out
G0 Z[(StartZ)] /-- Move to Starting Z
G28 (NumPasses) = [(NumPasses) + 1]
G27 #NextCut

#FinishedTaper
/-- Perform Final Pass
G1 X(StartX) Z(StartZ)
G1 X(EndX)
G1 X(StartX) Z(EndZ)

M02 /-- End of Program

```



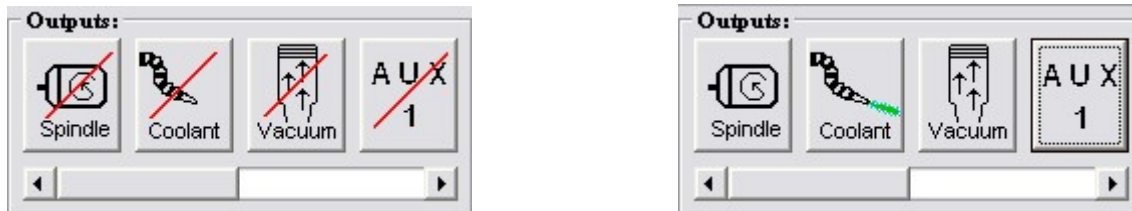
Customizing TurnMaster Pro

This section will instruct you on how to customize **TurnMaster Pro 2014**.

TurnMaster Pro 2014 is easily customizable. You can change the Output icons to reflect your machining needs.

Customizing Output Icons:

The following graphic is the Outputs menu with the default icons used.



In the **TurnMaster Pro** home directory there are 4 output icons in standard Windows Icon files (.ico)

out1-on.ico
out1-off.ico
out2-on.ico
out2-off.ico
out3-on.ico
out3-off.ico
out4-on.ico
out4-off.ico

Outputs 1, 2, 3 and 4 are programmable.

These standard Window Icon files must meet the following requirements:

Size: 48X48

Background: Transparent

Once you have created an Icon that meets the above requirements, you can save it in the **TurnMaster Pro 2014** home directory with the name that corresponds to the output position you wish to change. (As listed above)

There are many Freeware / Shareware icon editor / creators on the net today. Anyone that runs under Windows 9.x should be capable of creating these icons.

You can utilize a 3rd party image editor to create these files. There are also many shareware / freeware image editors on the net that would be capable of these simple graphical operations.

MN400 Controller Information

The MN400 controller card is an external motion control card that uses either a serial or USB port to communicate with the PC. This allows for the use of the MN400 controller board on any computer that has a serial or USB port. With this controller, you can now use **TurnMaster Pro** with Windows ME/2000/XP/Vista and/or a laptop computer.

Within this version of **TurnMaster Pro**, the MN400 can be activated and used to machine part programs, jog the machine, and do almost everything the OptiStep/QuickPhase and the Instep Library can do with the exception of utilizing Encoder feedback.

The MN400 is programmed to use COM ports at 9600 BPS. These parameters are adjustable.

**** NOTE:** If after connecting the MN400 to the serial or USB port on the back of your computer and choosing MN400 in the Machine Parameters, you are unable to jog, switch the cable to another available serial or USB port and try again.

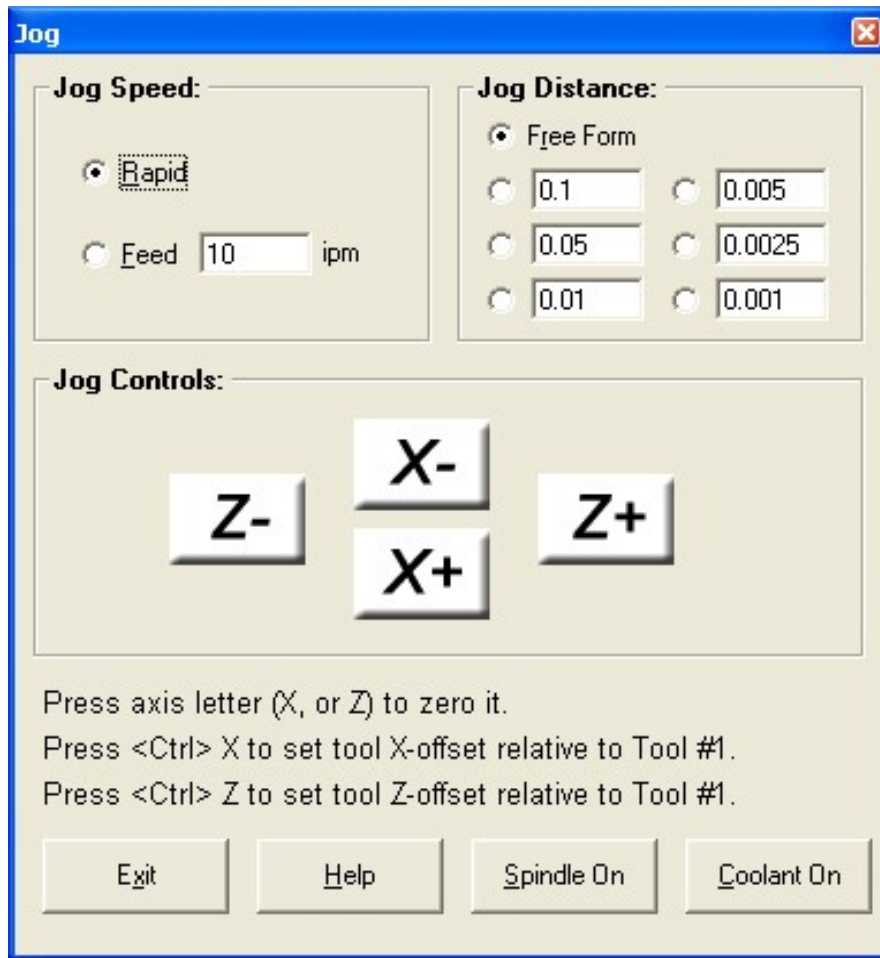
All commands work the same in **TurnMaster Pro** when using the MN400 except the jogging feature. The differences are described below.

Although the operation of the MN400 is extremely close to the operation of the Instep Library, the MN400 does have some small differences. When running a program with the MN400, you do not experience the "Lockout" that occurs when using Instep and either the OptiStep or QuickPhase cards. The MN400 is a more advanced controller that accepts commands directly from the software through a serial port, giving you more control over moves and program operation.

You may experience some delay when reading counters, when limits are hit or when running special operations such as Diagnostics, Home to Limits and Reprogram Limit Sensors. These delays are minor and are usually less than 1 second in length.

Jogging with the MN400 controller

With the MN400 motion controller board, you can now jog your machine with ease using the mouse. The graphic below shows the MN400 Jog window.



Each axis has its own + and - buttons organized in an easy to use fashion. The large white buttons contain the axis letter and the direction. To manually jog the machine, make sure Free Form is selected in the Jog Style frame and place your mouse cursor over one of the white buttons. Click and hold the mouse button down. The MN400 will jog the corresponding axis continuously until you release the mouse button.

You can also dispense Measured Moves by selecting Measured Moves in the Jog Style frame. Now, each click of one of the axis buttons will dispense the exact number of steps needed to move the Measured amount specified in the frame in the top left corner. To specify a custom distance, select 'Other' and enter the distance you wish to move in inches/mm. Now when you press any of the axis buttons, the machine will jog the distance specified.

The Jog Speeds frame allows the change of the jog speed based on either the MaxSpeed, defined under Machine Parameters, or by selecting feedrate and entering a value in inches per minute (ipm). For instance, to jog at a rate of 10 inches per minute, the user would select FeedRate and enter the number 10 in the field. Now when jogging any axis, the motors will move at 10 inches per minute.

For quick help while in the MN400 jog window, click the **Help** button at the bottom of the screen. This will display a message window giving you basic instructions for jogging with the MN400 controller.

Appendix A

Keyboard Scan Code Reference

The numbers to the right of the keys are the scan (or "make") codes.
These codes are used for programming the active key for each tool jog direction.

F1 – 59	0 – 11	S – 31	Left ALT – 56	Keypad 1 – 79
F2 – 60	A – 30	T – 20	Left SHIFT – 42	Keypad 2 – 80
F3 – 61	B – 48	U – 22	Right SHIFT – 54	Keypad 3 – 81
F4 – 62	C – 46	V – 47	Caps Lock – 58	Keypad 4 – 75
F5 – 63	D – 32	W – 17	SCR LOCK – 70	Keypad 5 – 76
F6 – 64	E – 18	X – 45	NUM LOCK – 69	Keypad 6 – 77
F7 – 65	F – 33	Y – 21	DEL – 83	Keypad 7 – 71
F8 – 66	G – 34	Z – 44	INS – 82	Keypad 8 – 72
F9 – 67	H – 35	- – 12	TAB – 15	Keypad 9 – 73
F10 – 68	I – 23	= – 13	CTRL – 29	
1 – 2	J – 36	\ – 43	ESC – 1	
2 – 3	K – 37	[– 26	SpaceBar – 57	
3 – 4	L – 38] – 27	BKSPC – 14	
4 – 5	M – 50	; – 39	Keypad + – 78	
5 – 6	N – 49	“ – 40	Keypad - – 74	
6 – 7	O – 24	- – 28	Keypad * – 55	
7 – 8	P – 25	, – 51		
8 – 9	Q – 16	. – 52		
9 – 10	R – 19	/ – 53		

Appendix B

Metacommand Reference

Metacommands are special commands **found in comment lines** that affect the **setup** when the **CNC program file** is loaded.

If the same **metacommand** appears more than once in the same file, the last one found is used.

Below is a list of the **metacommands** available along with the **proper syntax: data** is a **decimal number** and **flag** is either **1** or **-1**. (**1** is used for **LEFT** or **DOWN** and **-1** is used for **RIGHT** or **UP**).

Some of the commands can only be used with the constants listed next to them.

___ {Fixture}:CHUCK or CENTERS ___ {Units}: INCH or MM
___ {Part Length}:*data* ___ {O.D.}:*data* ___ {I.D.}:*data*
___ {X Start}:*data* ___ {Z Start}:*data*
___ {Z Origin Zero}:*flag*

Use these commands in each part program to **automatically setup TurnMaster Pro!**

Example:

```
/ {Fixture}: CHUCK      {Units}: INCH  
/ {Part Length}: 2.0000    {O.D.}: 1.0000      {I.D.}: 0  
/ {X Start}: .550            {Z Start}: 0.050  
/ {Z Origin Zero}:-1
```

This set of **metacommands** will configure **TurnMaster Pro** to **Inch mode** and set the **fixturing method** to a **lathe chuck**.

The material size will be set to a 1" diameter solid round stock, 2" long.

The origin will be placed to the right side of the material and the tool **start position** will be set at 0.05" outside and **0.05" to the right of the material.**

Appendix C

Wiring Diagram for External Cutting Speed Control Option

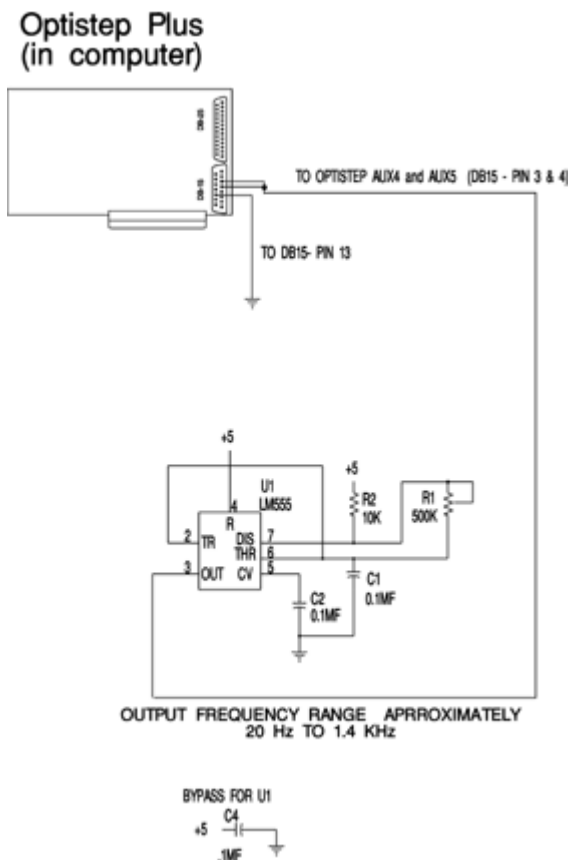
Connect an external pulse source, as in the example below, to the OptiStep Plus controller to use the External Cutting Speed Control option.

To enable this option, select External Cut Speed Control under Options-Position & Feed Speeds.

In this example, a free running oscillator circuit is sending a step rate to the OptiStep Plus PC. The step rate can be varied from approximately 20 Hz to 1400 Hz by adjusting R1 (500 K OHM potentiometer)

NOTE: This application is intended for spindle synchronous feeds. To manually override the feedrates, an OptiStep Plus FR is required.

Example:



Appendix D

Threading Example Using the G33 Command

The **G33** command is used to cut internal and external, left and right handed threads. In this example, an American Standard type 3/4-10 NC x 1-1/4 right-handed bolt will be cut. Below is the **G33** statement that is used:

```
G 33 X-0.065 Z-1.1 K.1 Q30
```

The **X** parameter value is the depth of the thread. The (-) sign indicates that the tool will travel in the **X**- direction to remove material.

Prior to the **G33** command being executed, the tool's **X**-coordinate should be positioned 0.050" above the material.

The **Z** parameter value is the distance the tool will travel each cutting pass. The (-) sign indicates that the tool will travel in the **Z**- direction thus producing a right-handed thread. Change the sign to (+) to produce a left-handed thread.

The **K** parameter value is the pitch of the thread. It can be found by dividing the threads per inch (TPI) into 1.

Example: The pitch of a 5 TPI thread can be found: $\text{pitch} = 1 / 5 = 0.2"$

The **Q** parameter value is the number of passes that will be made to cut the depth of the thread. This parameter can be replaced with an **I** parameter which specifies the depth of each cut.

NOTE: The following requirement must be met in order to cut threads using **TurnMaster Pro**.

$Z\text{-Axis TPI} \times \text{Steps/Rev} < \text{Encoder Res.} \times \text{Min. TPI to be cut}$

where **Z-Axis TPI** is the threads per inch of your lead screw, **Steps/Rev** is the resolution of the stepper motor driving the axis, **Min. TPI to be cut** is the minimum number of threads per inch to be machined, and **Encoder Res** is the resolution of the encoder on the spindle shaft.

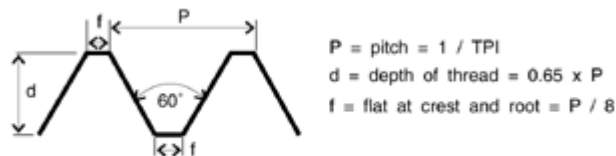
Example:

What encoder resolution is required to cut a 5 TPI thread with a 20 TPI lead screw and a 400 steps/rev stepper motor ?

$(20 \text{ TPI}) \times (400 \text{ Steps/Rev}) < \text{Encoder Res} \times (5 \text{ TPI})$

so: $\text{Encoder Res} > 1600 \text{ pulses / Rev}$

Select an encoder that has OPEN COLLECTOR outputs and a $V_{cc} = +5 \text{ Vdc}$.

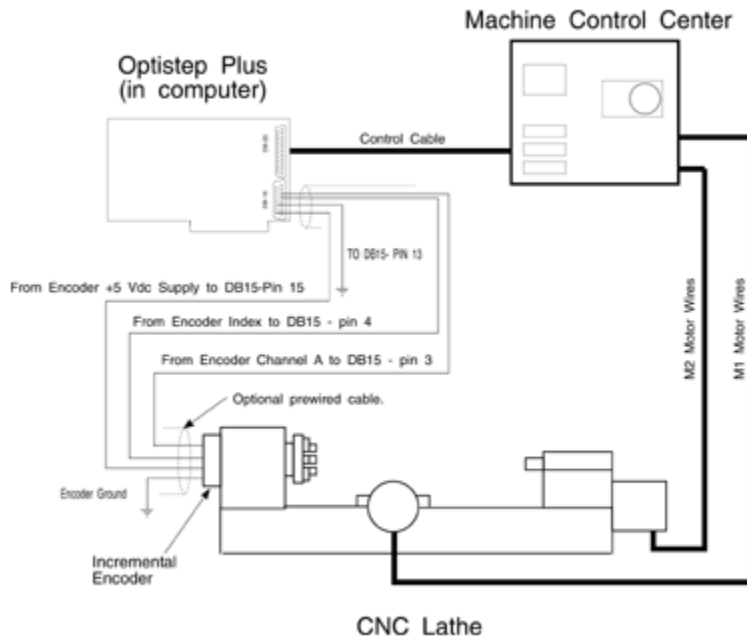


The American National form of thread

Wiring Diagram for Spindle Motor Encoder used for Threading

The lathe spindle motor is equipped with an incremental rotary encoder. The encoder is sending step and index pulses to the OptiStep-Plus so that threading can be accomplished.

The encoder has OPEN COLLECTOR outputs and a supply voltage of +5 Vdc.



Appendix E

Using the configuration File BENCHTOP.TPS

The purpose of this **appendix** is to define each parameter in the **BENCHTOP.TPS file**. The **BENCHTOP.TPS file** must be in the same subdirectory as the **TMPRO.DEF file**. This is usually your **TurnMaster Pro** home directory (i.e. **C:\Program Files\TMPFW**).

The **configuration file** for **TurnMaster Pro** has changed from the previous **DOS Version**. The new configuration system is designed to allow the user to view and edit the file from within **TurnMaster Pro 2014** without cryptic DOS commands or opening any kind of external program and endlessly searching your hard drive for the correct file. Although, if the need arises to open this file, we have added detailed descriptions for each attribute of your software and of your machine for easy configuration and/or troubleshooting.

Note: This listing is just a sample **TPS** file included with the installation package. Other **TPS** files included are: **BenchtopMetric.TPS, 1236.TPS, 1236MS.TPS, 1236 MN400.TPS, 1236MS MN400.TPS**. The values listed may be different for your application.

These attributes are displayed, in order, below.

Version = 2.0
Card address = 592
Card type = OPTISTEP PLUS
Max depth per pass = 1
Maximum unramped speed = 1000
Maximum speed = 1200
Acceleration rate = 5000
Fast jog speed = 800
Slow jog speed = 200
External sync active = 0
X screw pitch = 20
X steps in rev = 400
Z screw pitch = 20
Z steps in rev = 400
X plus = 72
X minus = 80
Z minus = 77
Z plus = 75
X origin = 1
Z origin = -1
X polarity = -1
Z polarity = 1
Part x dmr = 2
Part i dmr = .4
Part dim z = .4
Tool Origin = BOTTOM

Machine scale = 1
Using chuck = 0
X start pos = 0
Z start pos = 0
X limit pos = -2.522901
Z limit pos = 3.0623
X limit dir = -1
Z limit dir = 1
Units = I
X will test = 0
Z will test = 0
Diag Start Speed = 1000
Diag Speed Increment = 200
Diag Acceleration = 5000
X Distance = .4
Z Distance = 4
Tolerance = 0
Default tool = 1
Xbacklash = 0
Zbacklash = 0
Xbacklash Direction = 1
Zbacklash Direction = 1
Encoder Tolerance = 12
X Encoder Active = 0
Z Encoder Active = 0
Encoder Auto Correct = 0
Jog Tap Steps = 1
Feedhold Mode = 0
Multiplex = 4
OptiTracker Address = 0
Spindle Pulse Mode = 0
Axis #3 Type =
Axis #4 Type =
Steps Per Deg RPM = 10
Home Speed Pot Steps = 0

Output number = 1
Output name = preheat
Delay after on = 5
Delay after off = 5

**** Each Output utilizes 4 lines for its particular attributes. There are 8 programmable Outputs. ****

Tool Number = 1
Tool Name = left hand al
Tool Picture Filename = c:\program files\turn master pro\al.gif
X Tool Offset = 0
Z Tool Offset = 0
Tool Nose Radius = .005
x cut point = 2
y cut point = 2
Tool Rotation = 0

**** Each Tool utilizes 9 lines for its particular attributes.**

**** NOTE:** The **configuration file** will continue past this point depending on the number of tools defined in the "**Tool Chest**". There are a total of 99 tools available.

Below is an explanation of each attribute and Valid Ranges these attributes will accept.

Version

This parameter specifies the current **TurnMaster Pro 2014** version number.

Card Address

Valid Range: Specific Card Address (See card instructions for valid addressing)

This parameter specifies the card base address. (**Factory Address = 592**)

Card Type

Valid entries: QuickPhase, OptiStep, OptiStep Plus, MN400

This parameter specifies which card type is be used to operate the machine.

Max Depth Per Pass

This parameter specifies the maximum amount of material removed in each pass.

Max. Unramped Speed

This parameter specifies the maximum instantaneous speed at which the machine can be moved without acceleration. This speed is in steps per second and can easily be converted from inches per minute by using the following formula:

$$\frac{\text{inches}}{\text{min}} = \left(\frac{\# \text{ steps}}{\text{sec}} \right) * \left(\frac{1}{\text{screw pitch}} \right) * \left(\frac{1}{\# \text{ steps/rev}} \right) * \left(\frac{60 \text{ sec}}{\text{min}} \right)$$

Max Speed

This parameter specifies the **maximum speed** in which the machine can be moved.

Acceleration Rate

This is the rate at which the speed will increase from the **Max UnRamped** speed to the **Max Ramped** speed.

Fast Jog Speed

This parameter specifies the speed at which the machine will **Jog in Fast Mode**. This parameter should be set to the same speed as the **Max Unramped** speed.

Slow Jog Speed

This parameter specifies the speed at which the machine will **Jog in Slow Mode**. Set the speed that can be used for fine positioning. A good value to start with is **200**. (**Adjust the value as needed**).

External Sync. Active

This parameter specifies the signal for wire **EDM** machining. This parameter is **normally set to 0**.

X, Z Screw Pitch

This parameter specifies the **Teeth/Turns Per Inch (TPI)** of the lead screw.

X, Z Steps In Rev

This parameter specifies the number of steps the motor needs to complete 1 full revolution. (**MicroKinetics Motors are typically: Fullstepping = 200 and Halfstepping = 400**)

X Plus / X Minus

These parameters reflect the **Keyboard Scan Codes** that will control the **X Plus** and **X Minus** jog control keys. (**See Appendix A for Keyboard Scan Code information**)

Z Plus / Z Minus

These parameters reflect the **Keyboard Scan Codes** that will control the **Z Plus** and **Z Minus** jog control keys. (**See Appendix A for Keyboard Scan Code information**)

Z Origin

This parameter specifies whether the **Z** origin is to the right of the part (**Z Origin = -1** or **Z Origin = RIGHT**) or to the left of the part (**Z Origin = 1** or **Z Origin = LEFT**).

X, Z Polarity

Valid Range: NORMAL or REVERSE

These parameters specify the **Polarity** for each **Axis**. **NORMAL Axis Polarity** is defined as when the stepper motor rotates **CCW**, as viewed from the motor end. **REVERSE Axis Polarity** is defined as when the stepper motor rotates **CW**.

Part x dmr, Part i dmr, Part dim z

Part x dmr specifies the outside diameter (**X** dimension) of the part. **Part i dmr** specifies the inside diameter of the part. **Part dim z** specifies the length (**Z** dimension) of the part.

Machine Scale

This parameter specifies the **Scaling Factor**.

The scaling factor is used to scale the size of the actual part of an existing part program without changing any code.

Example: If a part program turned a part that is 2.0" square with the scale factor equal to 1, then with a scale factor equal to 2, the part would be turned would be a 4.0" square.

X, Z Start Pos

These parameters specify the **X** and **Z** start position of the tool.

X, Z Limit Position

These parameters specify the **X**, **Y** & **Z** limit positions for the machine. Your machine **must** be equipped with a **limit sensor package** to utilize this feature.

X, Z Limit Direction

This parameter specifies the direction in which the motor should **home/reposition** to.

Units

Valid Range: "I" for Inches OR "M" for Metric

This parameter specifies the unit of measure to be used for all software and machining operations..

Origin Visible

X, Z Will Test

Valid Range: 1 or -1

These parameters specify which axes will be tested during the **Diagnostics Testing** of the machine.

Diag Start Speed

This parameter specifies the **Start Speed** of the machine during the **Diagnostics Testing**.

Diag Speed Increment

This parameter specifies the **Increase In Speed Per Test**.

Diag Acceleration

This parameter specifies the **Acceleration** for ramped moves in **Steps/Sec²**.

X, Z Distance

This parameter specifies how far to travel for **Diagnostic Testing**.

Tolerance

This parameter specifies the **Tolerance** for machining.

Example: A 1 inch square block with a tolerance of **+/- 0.005"** could be in the range of **0.995" to 1.005"** and still be within tolerance. **Total tolerance in this case is 0.010" - (0.005" for the plus and 0.005" for the minus)**

Default Tool

Valid Range: 0 - 99

This parameter specifies the default tool that **TurnMaster Pro** will load at program startup or when restoring defaults.

Xbacklash and Zbacklash

Valid Range: 0 to 0.9999

These parameters specify the amount of **backlash** in each of the **2 axes**. Enter the amount of **backlash** in the units specified in the **Options - Material Setup Parameters dialog window**.

Xbacklash and Zbacklash Direction

Valid Range: 1 or -1

These parameters specify the **uncompensated** tool direction in each of the 3 axes. Enter **1** for **positive** or **-1** for **negative backlash setup direction**.

Encoder Tolerance

Valid Range: # of steps

This parameter specifies acceptable error in number of encoder pulses.

Example: If **Encoder Tolerance = 6** then **TurnMaster Pro** would not flag a positioning error until the absolute position of the tool was 7 or more encoder pulses off.

If the encoder has **400 pulses per revolution** and the lead screw is **20 T.P.I.**, then this example would have a tolerance of **+/- 0.00075"**.

Xencoder, Zencoder Active

Valid Range: YES or NO

These parameters turn encoder feedback **ON** or **OFF**. Specifying **YES** turns **ON** encoder feedback for that axis (**closed loop mode**) and a **NO** turns it **OFF** (**open loop mode**).

Encoder Auto Correct

Valid Range: YES or NO

This parameter specifies the action **TurnMaster Pro** will take if a positioning error exists. Specifying **YES** will direct **TurnMaster Pro** to auto correct the error and **NO** will prompt the user to [**Abort**] or [**Correct**] and **Continue**.

Jog Tap Steps

Valid Range: # of steps

This parameter specifies the number of steps an axis will move when a **Jog Key** is **Tapped**.

Example: If **Jog Tap Steps = 8** then **TurnMaster Pro** will move any axis 8 steps when one of its respective **Jog Keys** is **Tapped**.

Feed Hold Mode

Valid Range: Yes or No

This parameter specifies the action **TurnMaster Pro** will take when the shield is opened. Specifying **NO** will stop machining, turn **OFF** the **Spindle Motor** and prompt the user to [**Abort**] or [**Continue**]. Specifying **YES** will simply pause machining (**spindle motor remains on**)

Multiplex Resolution

Valid Range: 1, 2 or 4

This parameter adjusts the **Interpolation Resolution**. With **higher resolution**, there is less vibration on diagonal moves due to **Velocity Ripple**. A value of **2** is suitable for most applications.

OptiTracker address

Valid Range: Decimal base address

This parameter specifies the base address of the **OptiTracker(TM) card** in the computer. This parameter is ignored if set to 0.

Spindle Pulse Mode

Valid Range: YES or NO

Setting Spindle Pulse mode to YES causes M03 spindle on (CW) command to generate a pulse on output #1, M04 (CCW) causes a pulse on Output #5, and M05 (Spindle OFF) causes a pulse on output #4. These pulses are active low for a period of 1 second for M03 and M04. The period for the OFF command is 0.25 seconds..

Axis #3, #4 type

Valid Range: SPINDLE

This parameter specifies what type the 4th axis is. Usually a rotary table.

Steps Per Deg RPM

Valid Range: 80-100

This parameter specifies the number of steps that the motor must make for the rotary table to rotate one full degree when using the 4th axis as a rotary axis.

Home Speed Pot Steps

Valid Range: Trial and Error

This parameter is used if the 4th axis is specified as SPINDLE. Home speed pot steps is the maximum number of steps it takes to move the speed control / knob motor from anywhere in the range back to zero.

Output Number

Valid Range: 1-5

This parameter specifies the output number.

Output Name

This parameter is a text description of the output. For example, on a torch machine a sample output name might be 'Preheat'

Delay after on

Valid Range: an integer specifying the delay in seconds

This parameter specifies the amount of time TurnMaster should delay after turning on an output, before proceeding. You may specify, in seconds, 3 places to the left and one to the right. (###.#)

Delay after off

Valid Range: an integer specifying the delay in seconds

This parameter specifies the amount of time TurnMaster should delay after turning off an output, before proceeding. You may specify, in seconds, 3 places to the left and one to the right. (###.#)

Tool Number

Valid Range: 1-99

This parameter specifies the tool number in the index.

Tool Name

This parameter specifies the unique name for the tool. This parameter is displayed in the Tool Geometry window when the tool is selected.

Tool Picture Filename

This parameter specifies the full path and filename for the image associated with the tool.

Example: C:\images\tool1.gif

Please refer to Customizing **TurnMaster Pro** for further information on image association.

X Tool Offset

This parameter specifies the tool's X offset.

Z Tool Offset

This parameter specifies the tool's Z offset.

Tool Nose Radius

This parameter specifies the radius of the tool's nose.

X Cut Point

This parameter specifies the tool's graphical X cut point.

Y Cut Point

This parameter specifies the tool's graphical Y cutpoint

Tool Rotation

This parameter specifies if the tool's graphic is rotated 90 degrees.

Appendix F

Using Machine Parameters as Read-Only Variables

Selected values entered in the **Machine Parameters** screen are available for use in your G-Code programs.

These values are similar to user-defined variables, in that they must be used with surrounding parenthesis. As with user-defined variables, they may be used in arithmetic expressions.

These values are **read-only** and cannot be set using a **G28** command.

For example:

```
G00 x[.5*(TM_PARTDIAMETER)] z[-1*(TM_PARTLENGTH)]      /-- top left of part
G01 z[1000 \ [(TM_ZSCREWPITCH)*(TM_ZSTEPSINREV)]]      /-- move 1000 steps
```

The following values are available:

<u>Variable Name</u>	<u>Value</u>
TM_PARTLENGTH	Z-dimension of the work area
TM_PARTDIAMETER	X-dimension of the work area
TM_CONTROLLER	1 - MN400, 2 - OPTISTEP PLUS, 3 - QUICKPHASE
TM_UNITS	1 - Inches (English) , 2 - Millimeters (Metric)
TM_MACHININGSCALE	Machining Scale (Default = 1)
TM_TRACKJOG	Track External Control Panel ? (Yes = 1, No = 0)
TM_CONTCONTOURING	Initially Use Continuous Contouring ? (Yes = 1, No = 0)
TM_MAXUNRAMPEDSPEED	Maximum initial axis speed (steps/sec)
TM_MAXRAMPEDSPEED	Maximum final axis speed (steps/sec)
TM_ACCELERATION	Axis acceleration (steps/sec ²)
TM_XORIGINZERO	X-origin (Top = 1, Bottom = 2)
TM_ZORIGINZERO	Z-origin (Right = -1, Left = 1)
TM_XSCREWPITCH	X-axis screw pitch (revolutions per inch)
TM_ZSCREWPITCH	Z-axis screw pitch (revolutions per inch)
TM_XSTEPXINREV	X-axis steps per revolution
TM_ZSTEPXINREV	Z-axis steps per revolution
TM_XPOLARITY	X-axis polarity (Normal = 1, Reverse = -1)
TM_ZPOLARITY	Z-axis polarity (Normal = 1, Reverse = -1)
TM_XBACKLASH	X-backlash value (Inches or MM)
TM_ZBACKLASH	Z-backlash value (Inches or MM)
TM_XBACKLASHDIR	X-backlash direction (Positive = 1, Negative = -1)
TM_ZBACKLASHDIR	Z-backlash direction (Positive = 1, Negative = -1)

TM_SERIALSPINDLE	Spindle (None = 0, Teco7300 = 1, Fluxmaster100 = 2)
TM_SPINDLECOMPORT	Serial Spindle Serial COM Port
TM_SPINDLEBAUD	Serial Spindle Serial Baud Rate
TM_SPINDLEACCDECTIME	Serial Spindle Acceleration/Deceleration Time (seconds)
TM_SPINDLEGEARRPM	Serial Spindle Gear RPM
TM_SPINDLETHREADSPEED	Serial Spindle Threading Start Speed
TM_AXIS3TYPE	Axis 3 (None = 0, Stepper = 1, AC Motor = 2)
TM_AXIS4TYPE	Axis 4 (None = 0, Rotary = 1, Spindle = 2)
TM_SPINDLERESETSTEPS	Axis 4 Spindle Speed Control Reset (steps)
TM_STEPSPERDEGRPM	Axis 4 Steps Per Degree RPM
TM_TOOLCHANGEPAUSE	Tool Change Pause ? (Yes = 1, No = 0)
TM_TOOLCHANGELOCK	Tool Change Lock ? (Yes = 1, No = 0)
TM_TOOLCHANGEOUTPUT	Tool Change Lock Output # (1-8)
TM_TOOLCHANGEINPOS	Tool Change In-Position Input ? (Yes = 1, No = 0)
TM_TOOLCHANGEINPUT	Tool Change In-Position Input # (1-8)
TM_TOOLCHANGEINPOSDelay	Tool Change In-Position Delay (secs)
TM_STEPSPERTOOLSTATION	Tool Changer Steps Per Tool Station
TM_REVSTEPSPERTOOL	Tool Changer Reverse Steps Per Tool Station
TM_UNIDIRECTIONALTURRET	Unidirectional Turret ? (Yes = 1, No = 0)
TM_TOOLSPERTURRET	Tools per Tool Turret
TM_TURRETUNRAMPEDSPEED	Tool Turret Max Unramped Speed (steps/sec)
TM_TURRETRAMPEDSPEED	Tool Turret Max Ramped Speed (steps/sec)
TM_TURRETACCEL	Tool Turret Acceleration (steps/sec ²)
TM_FEEDHOLD	Feedhold Mode ? (Yes = 1, No = 0)
TM_PULSEMODE	Spindle Pulse Mode ? (Yes = 1, No = 0)
TM_SPINDLESPEEDPAUSE	Tool Change Pause ? (Yes = 1, No = 0)
TM_USECHORDS	Use Chords for Arcs ? (Yes = 1, No = 0)
TM_CHORDSPERCIRCLE	Chords per Circle (4-720)
TM_MULTIPLEXRESOLUTION	MultiPlex Resolution
TM_EXTRACOMMDELAY	Extra Delay during MN400 Communications
TM_TOOLNUM	Active Tool # (1-99)
TM_TOOLRADIUS	Tool Radius for the Currently Selected Tool
TM_TOOLXOFFSET	Tool X-Offset for the Currently Selected Tool
TM_TOOLZOFFSET	Tool Z-Offset for the Currently Selected Tool
TM_TOOLRADIUSxx	Tool Radius for Tool # xx (where 1 <= xx >= 99)
TM_TOOLXOFFSETxx	Tool X-Offset for Tool # xx
TM_TOOLZOFFSETxx	Tool Z-Offset for Tool # xx