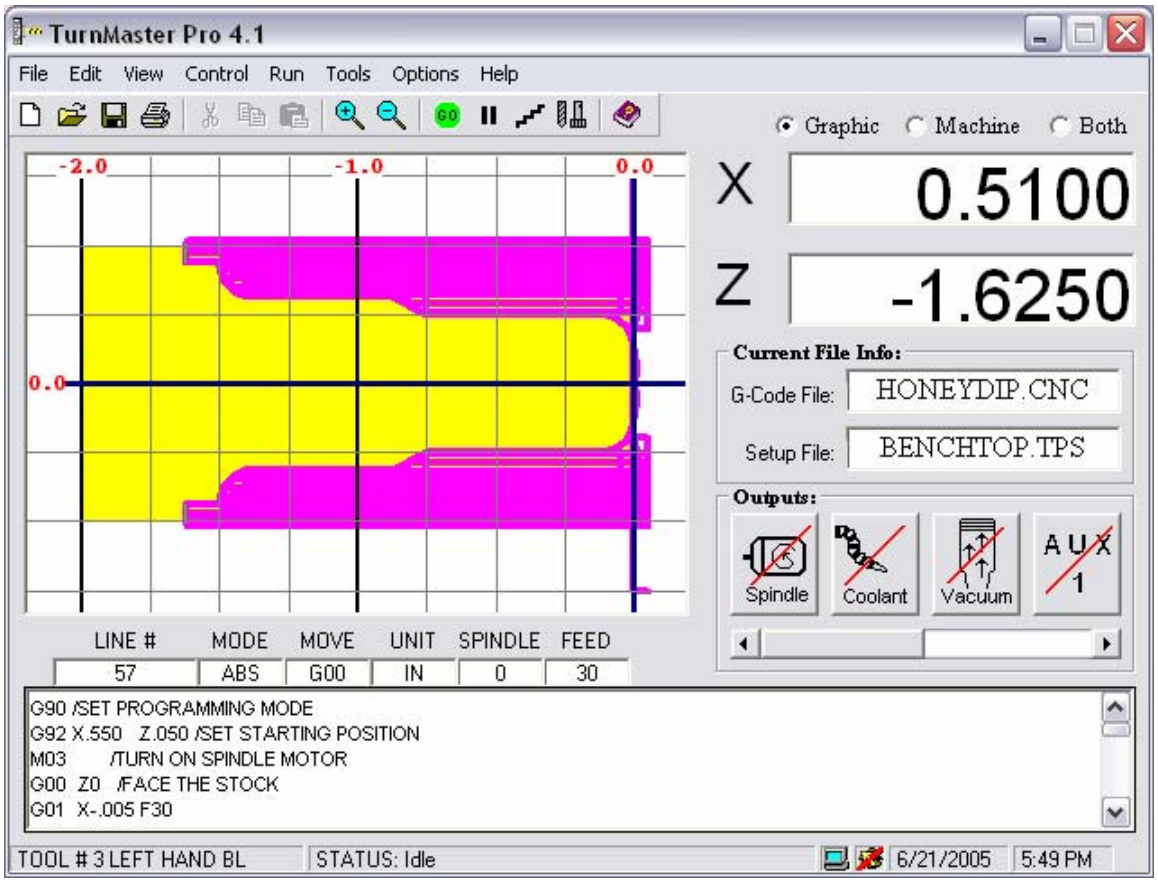
TurnMaster Pro for Windows

Turning Machine Control and CNC Program Verification
For Microsoft Windows ™ Based PC's
V4.2



Contents Quick Jump

Minimum System Requirements

- **An IBM PC or 100% compatible**
- **For OptiStep or QuickPhase control cards: Windows 98 (2nd edition) with Internet Explorer 5.5 installed and an available ISA slot**
- **For MN400: any version of the Windows operating system and a free serial port connection**
- **10-15 megs of free hard drive space**
- **Minimum 32 megs RAM (64 megs or higher recommended)**
- **Minimum 166mhz Processor (266mhz or higher recommended)**

Installing TurnMaster Pro for Windows

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

From the Windows Desktop:

1) Place the media containing TurnMaster Pro for Windows into a drive

2) Click **'START'; 'RUN'; 'BROWSE'**

3) Select the drive you inserted the media into, then select 'mmpfwv1.exe' and Click **'OK'**

This starts the Auto-Install process.

When this Dialog box appears on your desktop, click **'OK'.**

Next you will see the window below appear on your screen:

When this Dialog box appears on your desktop, click **'SETUP'.** This will Unzip all of the TurnMaster Pro files into a temporary directory and start the Setup program.

TurnMaster Pro will now be installed to your hard drive with the Standard Windows setup program. The next screen you will see has a Bluish/Black background. **(Shown below)**

Follow the prompts until Setup is complete.

***** 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 for Windows

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 an identical **TurnMaster Pro** Icon onto your desktop for quick, easy access.

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

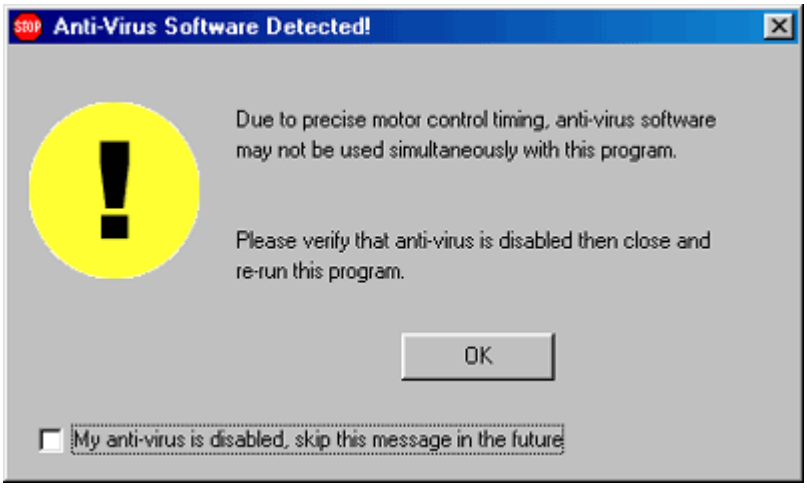
You will notice the evaluation message displayed on this screen until you 'Unlock' the software. This can be done by using the File Menu Bar at the top of the **TurnMaster Pro** Main Screen; clicking the word HELP, this will drop down a small sub-menu; click the option that says 'Unlock Software'

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 inquires 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 for Windows** 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 will see the following screen appear:



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.

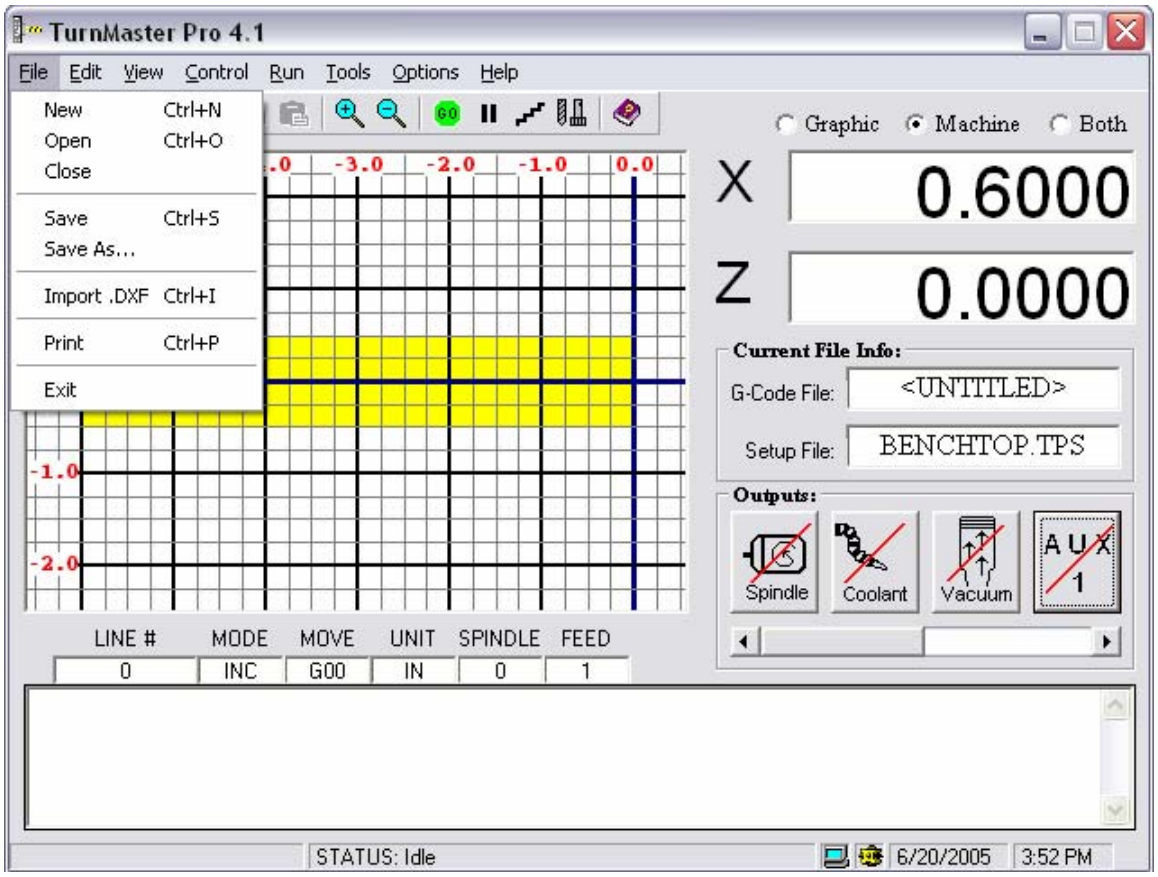
TurnMaster Pro only detects Norton Anti-Virus or McAfee VirusShield. If you are running another brand of Anti-Virus, this message will not be displayed. You will need to disable ANY virus protection, regardless of manufacturer before machining.

Getting Started with TurnMaster Pro for Windows

Thank You for purchasing **TurnMaster Pro for Windows 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.

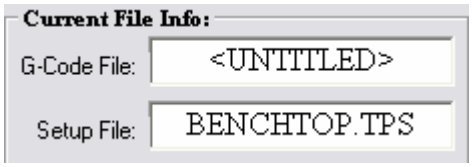
The following graphics show the Machine Parameter Tabs. The graphic on top demonstrates what the tab looks like when the **Mouse-over Hot Key** is **active**.

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** functions are also accessed with a **Mouse QuickSelect** click by left clicking and dragging the mouse over the area to zoom in on in the cutting window. Single clicking the right mouse button in this same window will bring your view back to the previous zoom level. Notice that when you move your mouse over the XY cutting window, the mouse pointer turns to a crosshair for precision zooming.

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.

4.1 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].**

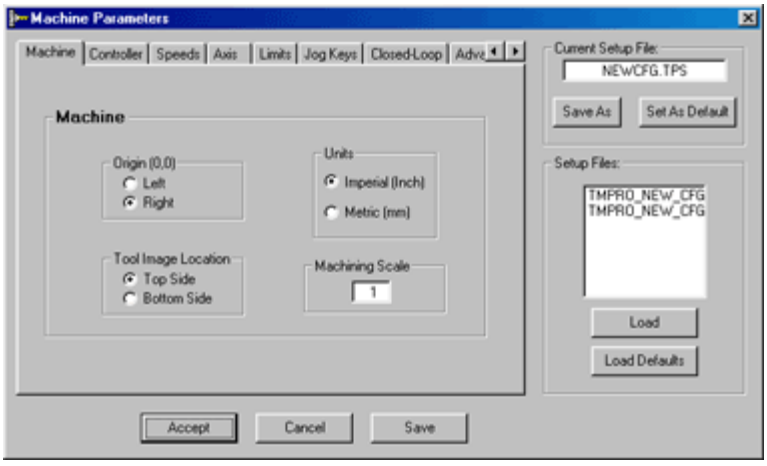
For Wood Tutorial:

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].**

4.2 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-Positions & Feed Speeds** Menu with the mouse or type **<ALT>-O M**.

1. Set the **Max Feed Speed** to the **Maximum Feedrate** that the stock material can be cut at. This speed is in steps per sec and can easily be converted from inches per minute by using the following formula:

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

   The default setting is **1500 Steps/Sec** and is adequate for **Soft Metal** and **Plastic**.

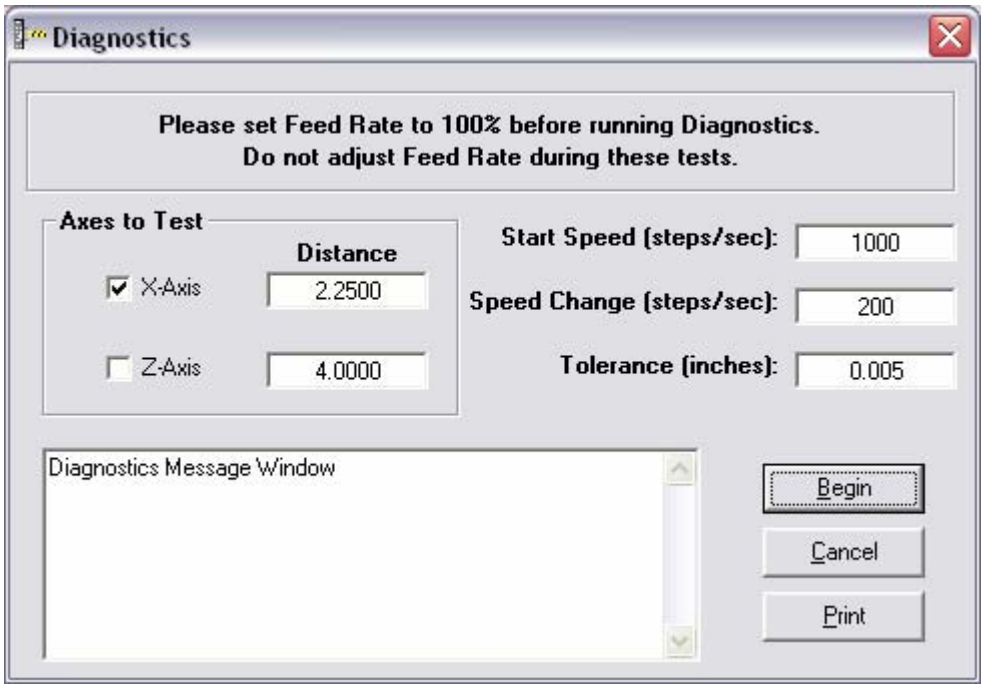
2. Set the **Rapid Traverse Speed** to the maximum speed at which your machine can move **reliably.** The default is set to **1800 Steps/Sec**. You may be able to set this at a much higher rate if you are using ballscrews or other low friction mechanical drives.

3. The ***Ramp Speed*** is an acceleration parameter and is usually set for **10,000 Steps/sec/sec** which is a good all-purpose value.

4. Set the **Fast Jog Speed** to the same speed as the **Max Feed Speed**.

5. Set the **Slow Jog Speed** to a speed that can be used for fine positioning. A good value to start with is 200. 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**.



4.3 Axis Control Parameters

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

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:

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

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**).

4.4 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.

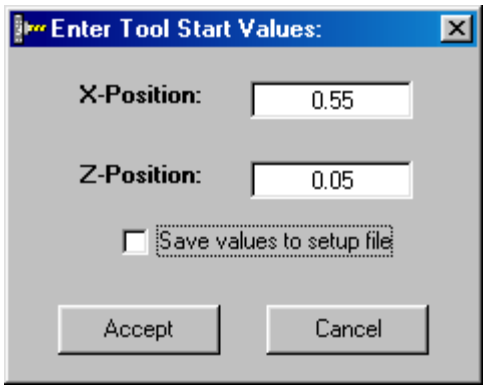
11 If you have limit switches installed on your machine, you can select Control- Reprogram Limit sensors to have it remember the start position as a reference.

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.

4.6 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 (see ***Figure 4.6.1***).
For the Wood Tutorial
Set it for: X-axis = 0.750"
Z-axis = 0"
to work with the tutorial (see ***Figure 4.6.2***).

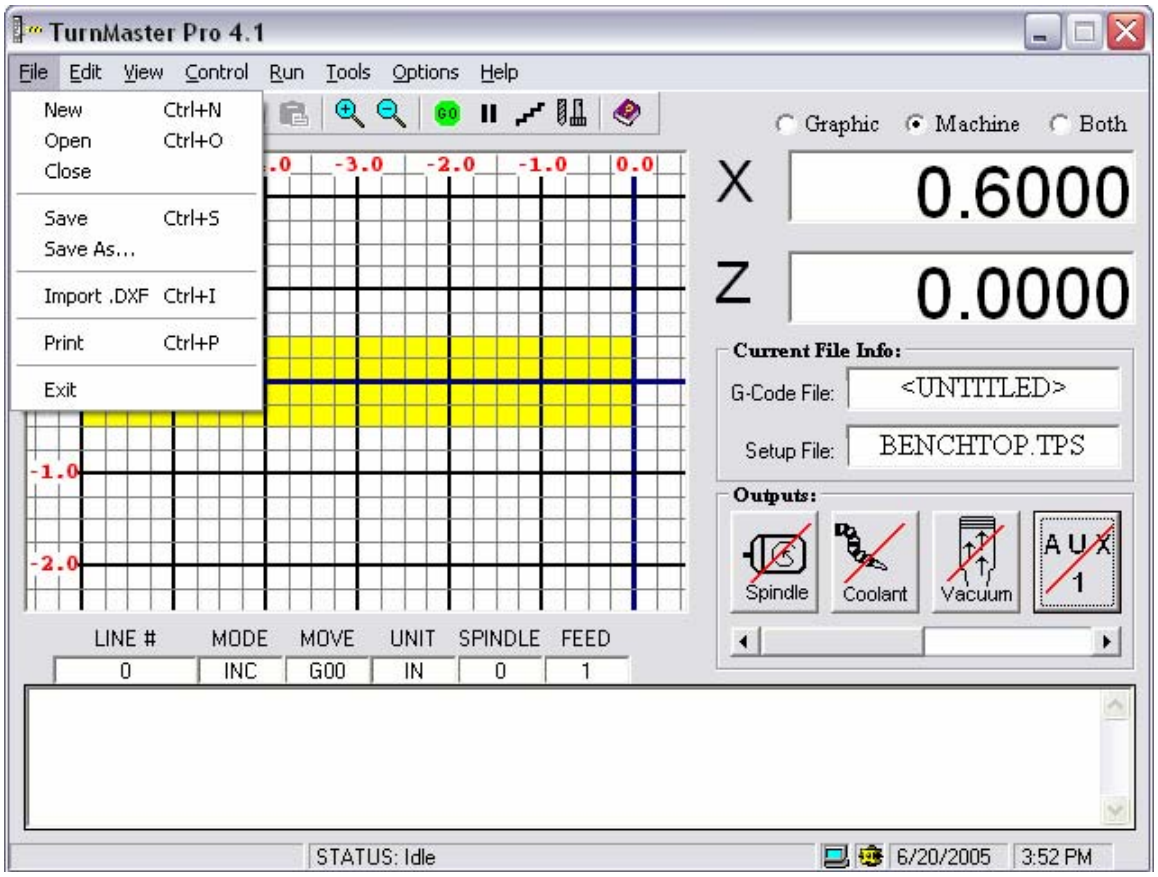
Choose **[OK]** when complete. These values may be changed to suit your specific needs.

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.

Menu Reference

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



File- *New*

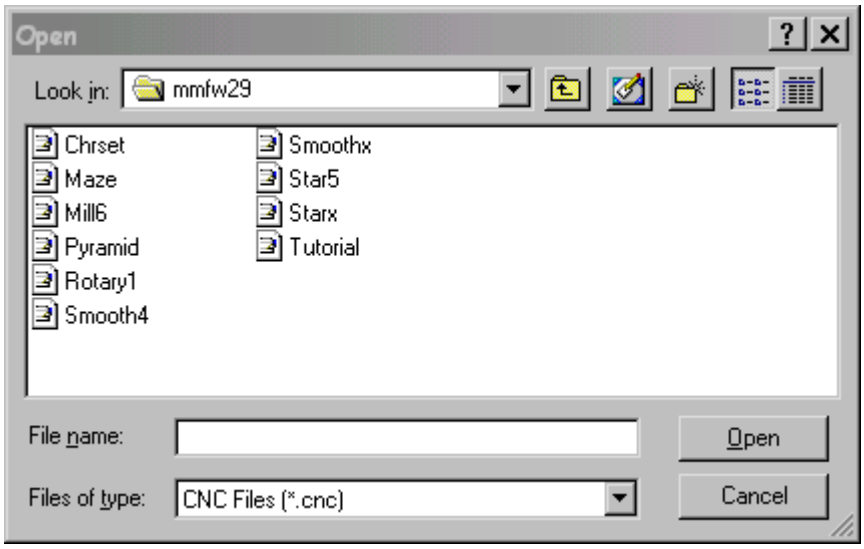
Clears the part program in memory and establishes the standard defaults for a new part program. The first icon in the Toolbar also corresponds to this menu command.

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

File- *Import .DXF*

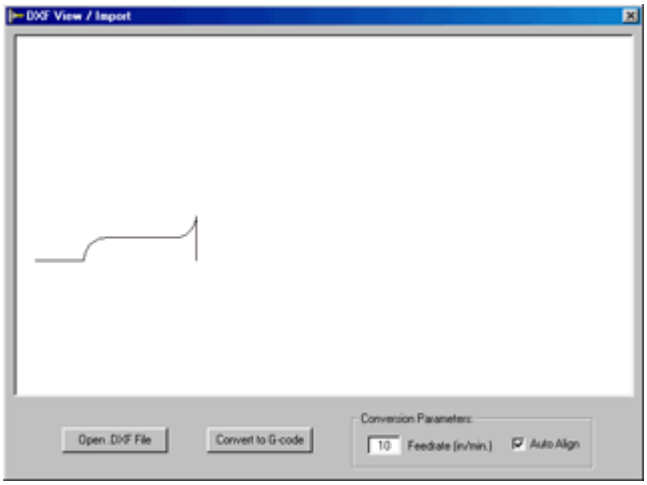
This menu command allows you to import **industry standard .DXF files up to version 15 (AutoCAD 2000 format**).This is the latest **AutoDesk** release.

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

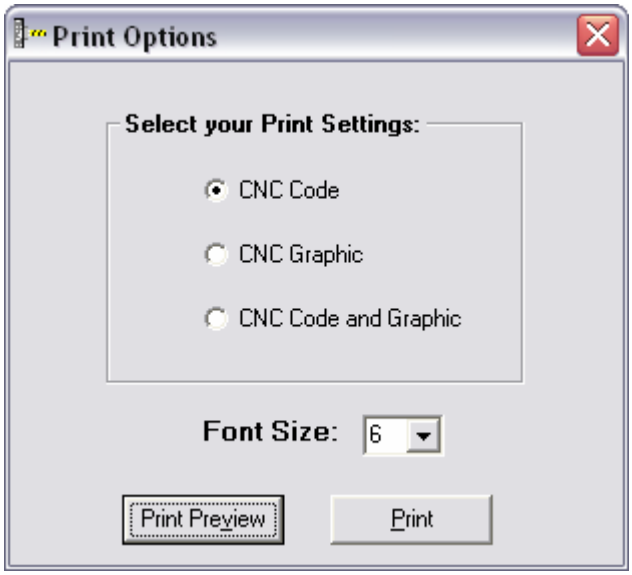
memory. If selected, when you return to the **main screen**, the program will be loaded.

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.

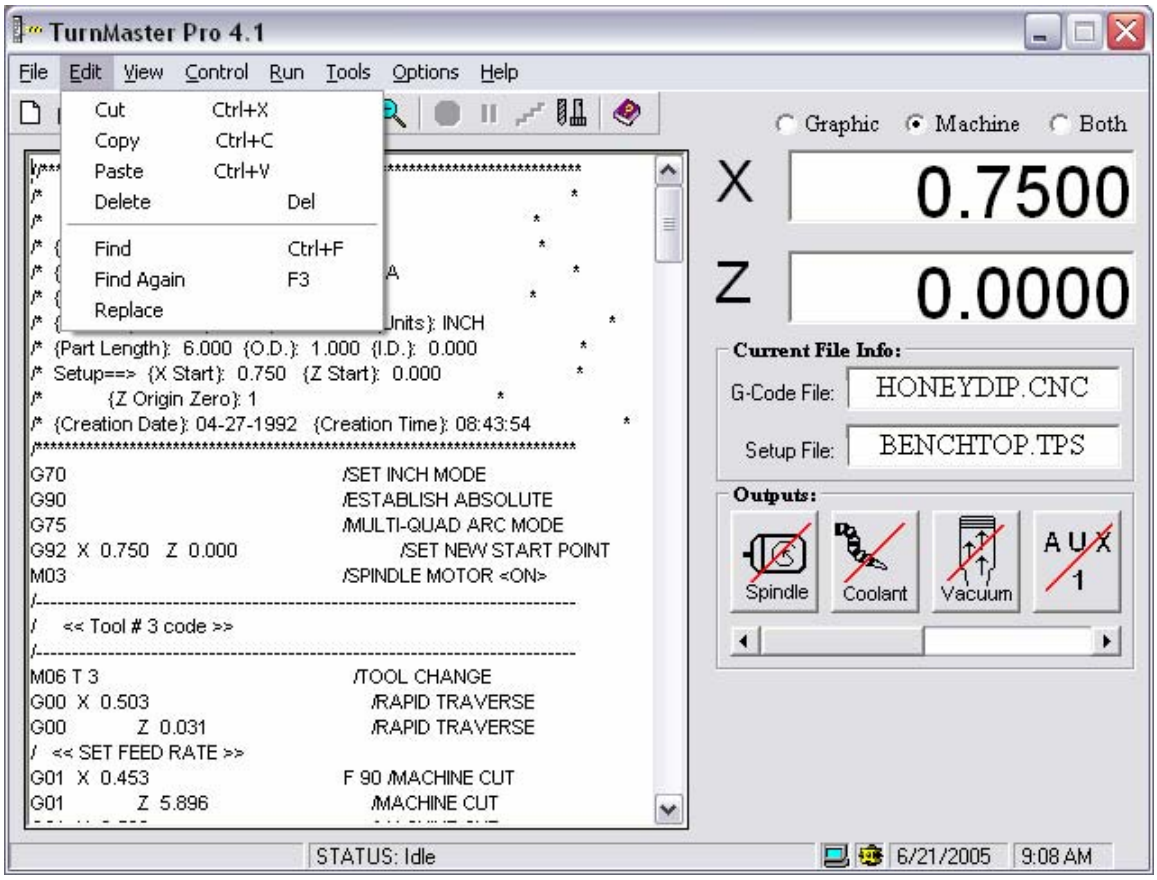


File- *Exit*

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.



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.

NOTE: This command is "grayed" out and disabled unless TurnMaster Pro is in CNC Edit Mode.

Edit- *Copy*

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

NOTE: This command is "grayed" out and disabled unless TurnMaster Pro is in CNC Edit Mode.**

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.

NOTE: This command is "grayed" out and disabled unless TurnMaster Pro is in CNC Edit Mode.**

Edit- *Delete*

Deletes the currently highlighted text permanently.

NOTE: This command is "grayed" out and disabled unless TurnMaster Pro is in CNC Edit Mode.**

Edit- *Find*

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

NOTE: This command is "grayed" out and disabled unless TurnMaster Pro is in CNC Edit Mode.**

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.

NOTE: This command is "grayed" out and disabled unless TurnMaster Pro is in CNC Edit Mode.**

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*

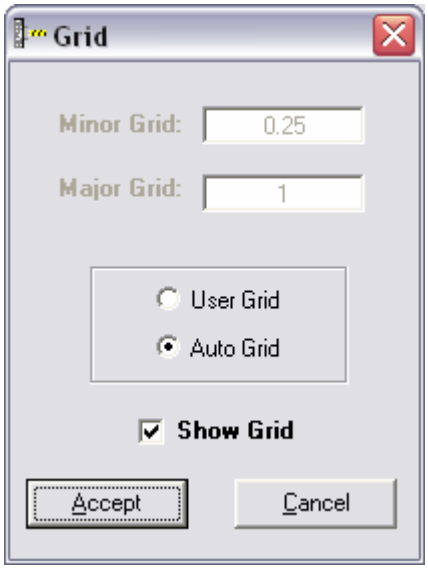
Allows the user to either enable or disable displaying the blue, crosshair Origin Marker over the tool in the XY viewport. When a check mark appears next to this function, it is enabled.

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*

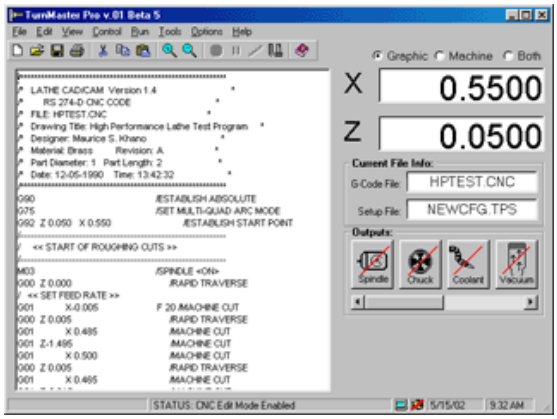
Displays the **XZ** Counters in **Full Screen Format**, while temporarily hiding all other controls. When a check mark appears next to this function, it is enabled. The graphic below represents the **Main screen in CNC Edit mode (left)** and the **Full Screen Count mode (right)**

View- *CNC Edit Mode*

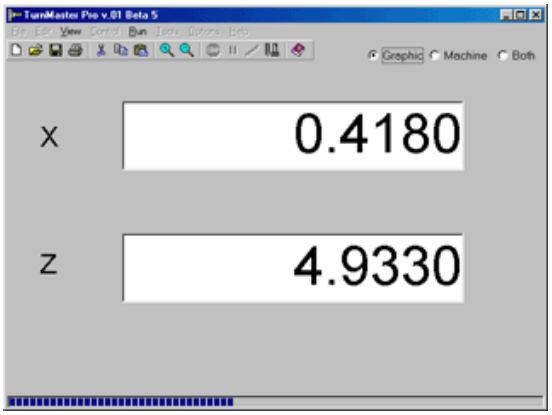
Turns on CNC text edit mode and displays the CNC text editor while temporarily hiding the cutting window. This allows the user to Edit any currently loaded program, or enter a part program manually. This works like any standard Text editor. When a check mark appears next to this function, it is enabled.

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



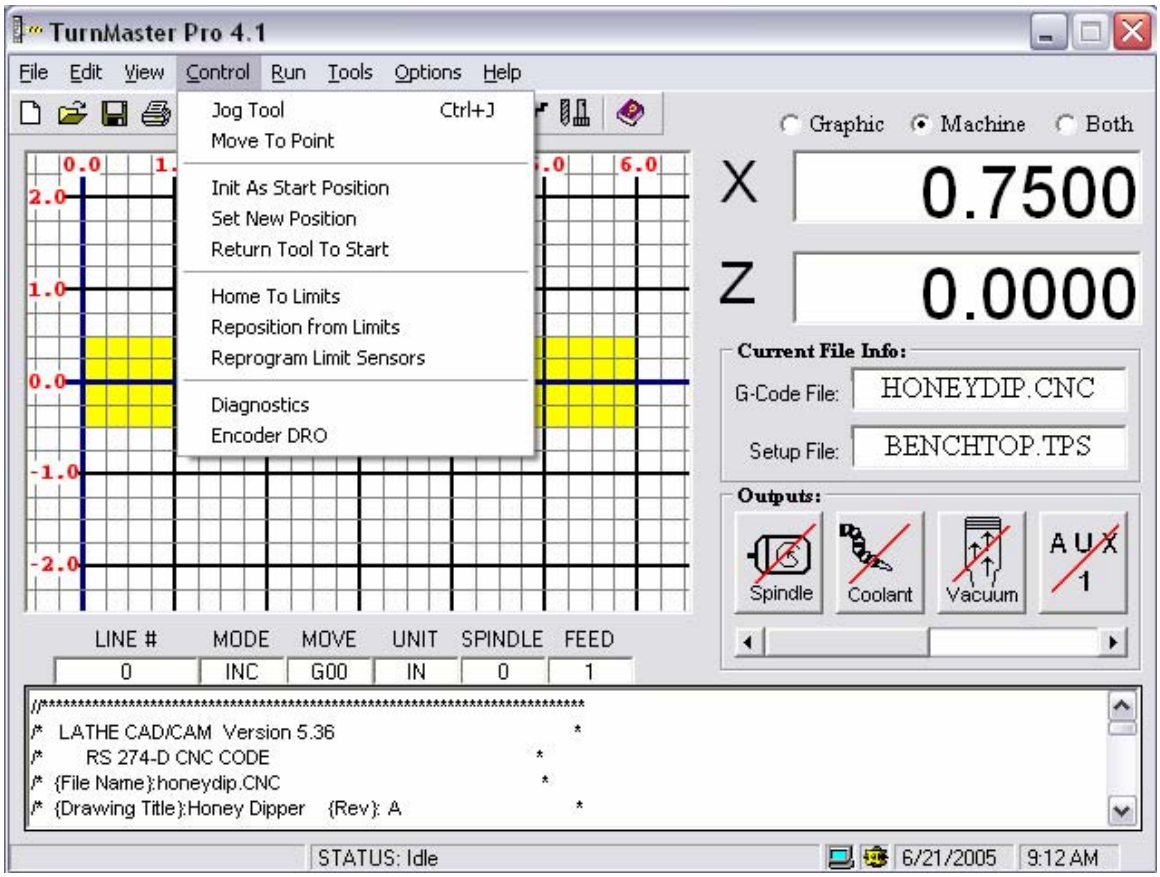
(These graphics have been shrunk for spacing purposes)

View- *Restore Screen*

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

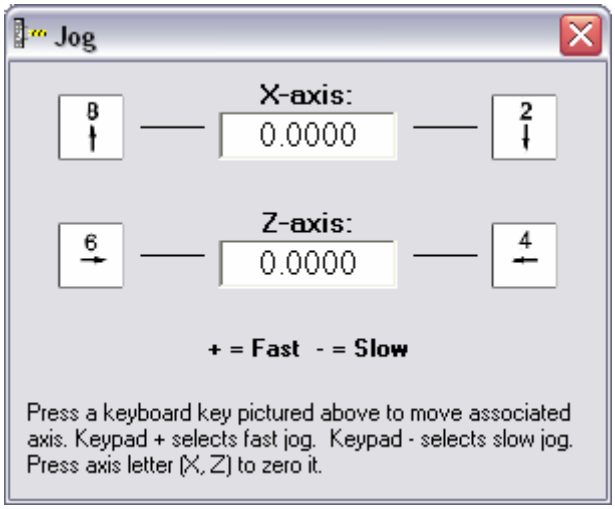
The values in the Material Size menu are applied here.

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*

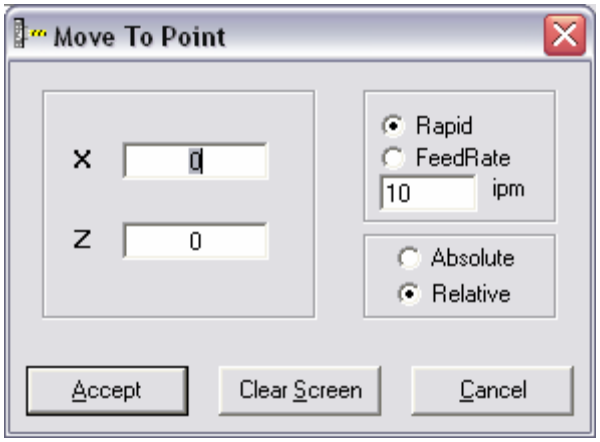
Allows the user to jog all axes on the machine. The absolute location of the tool is shown and updated when a jog key is released. A machine must be correctly connected to use this function. Pressing the + key enables fast jogging mode, while the - key enables slow jogging mode. Pressing the corresponding axis letter (X or Z) will zero that axis at its current location.



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.

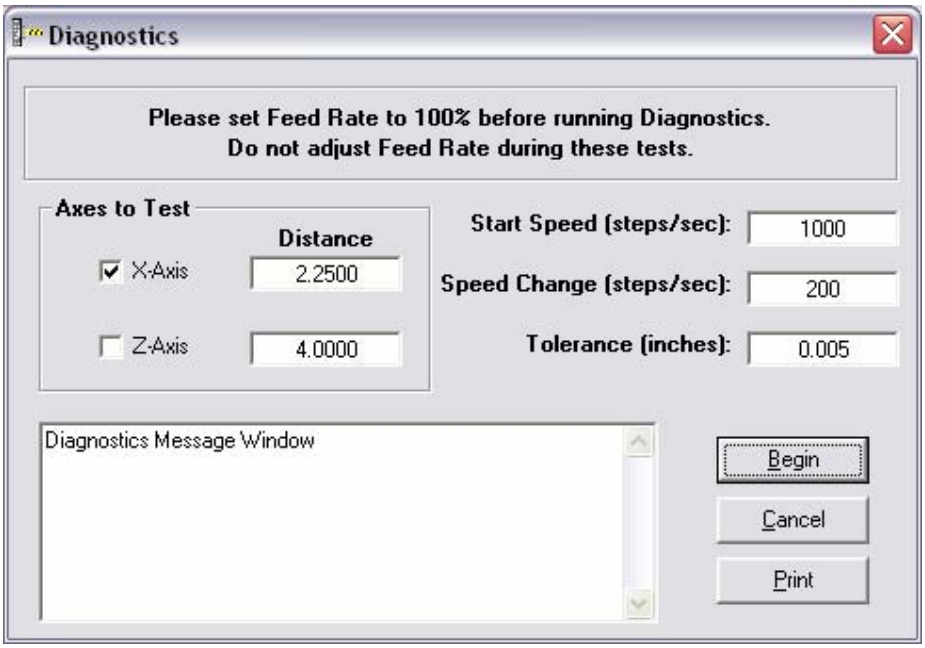
Control- *Reprogram Limit Sensors*

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

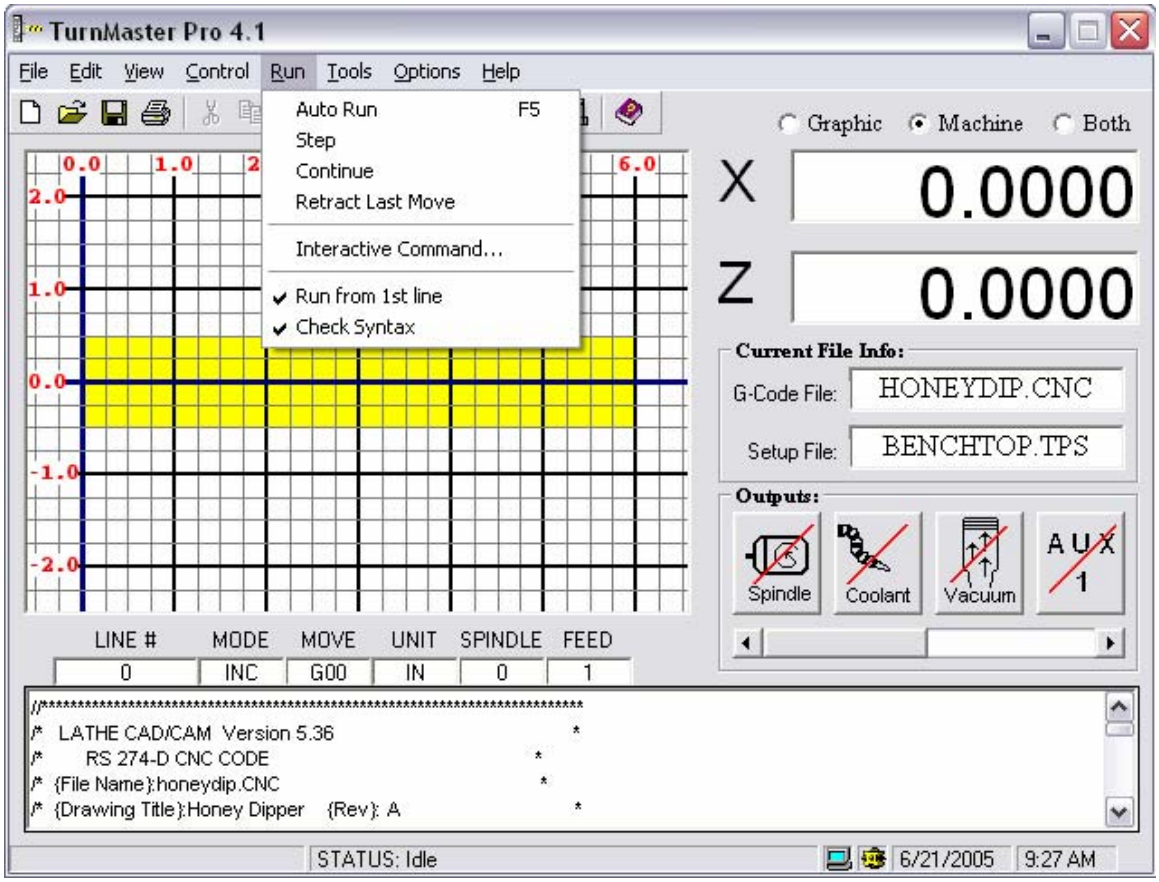
Control- *Diagnostics*

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 **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 tenth 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 eleventh 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.

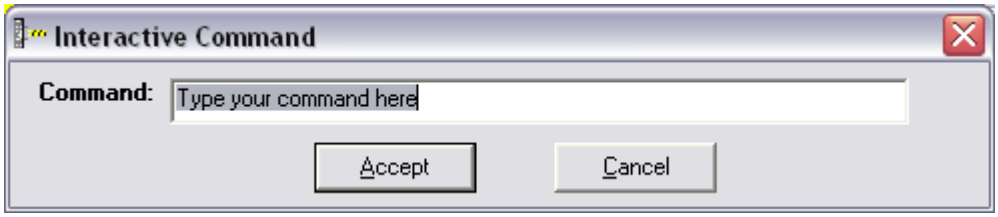
Run- *Retract Last Move*

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.

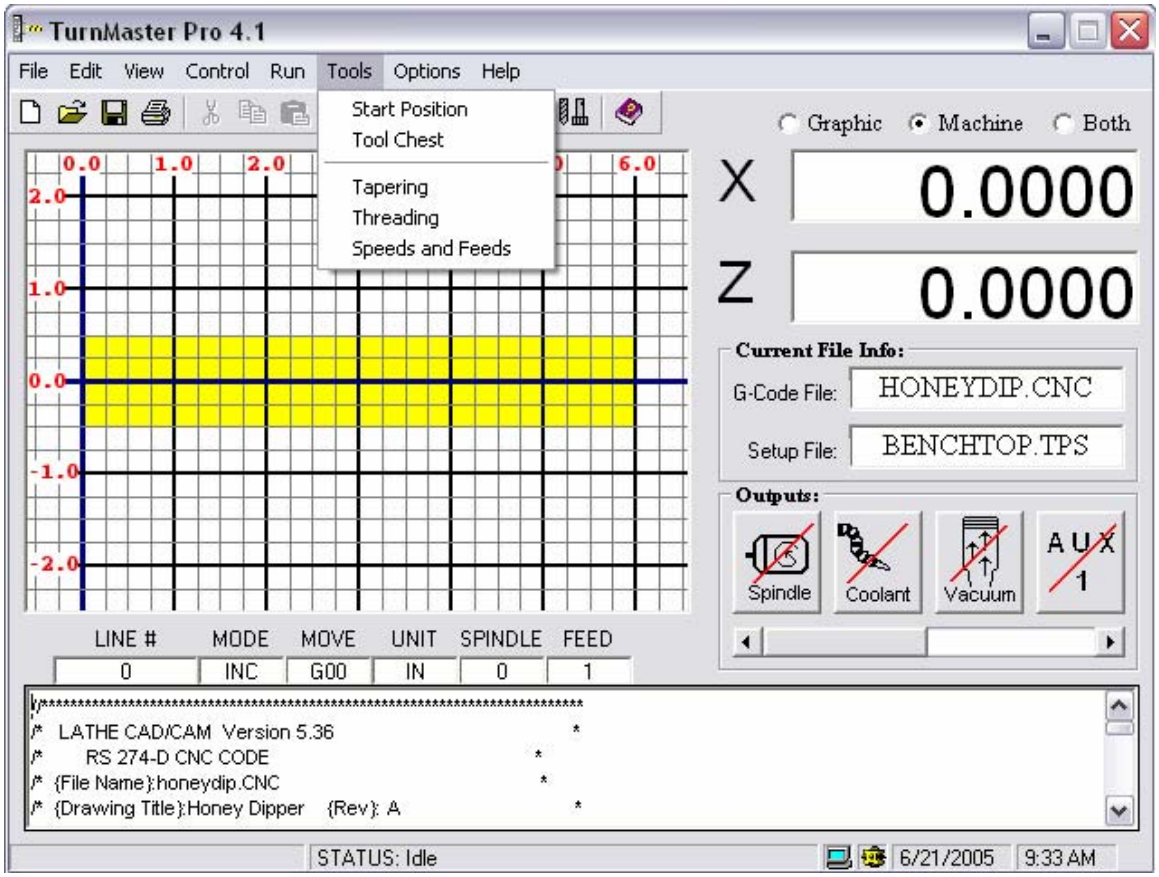
Run- *Check Syntax*

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 **Tools Menu** is used to specify the tool to use, its specific attributes, and to specify a starting position for the tool.

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

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



Tools- *Start Position*

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

Tools- *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 and select the **CutPoint** on that image for proper depiction in the graphical window on the main screen.

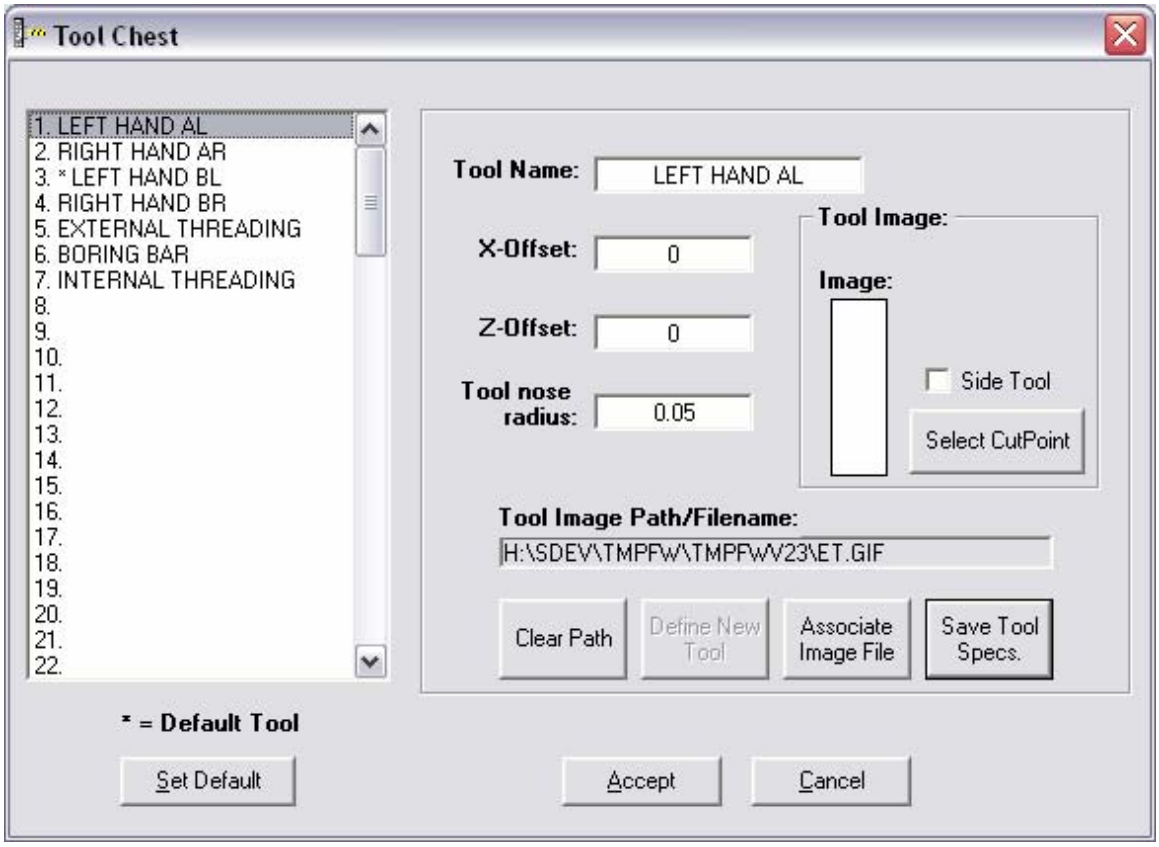
TurnMaster Pro comes with a set of tool images, which are contained in the TMPFW directory on your hard drive, usually under Program Files. After associating an image file, you will see it appear in the image box when you select the tool in the list on the left hand side of the window. Once all the tool information is visible, click the '**Select CutPoint**' button and click the exact spot on the tool image that you wish to be your cutpoint. A red dot will appear in the exact spot you chose. Click '**Save Tool Specs**' and you are finished.

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 vertical 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.

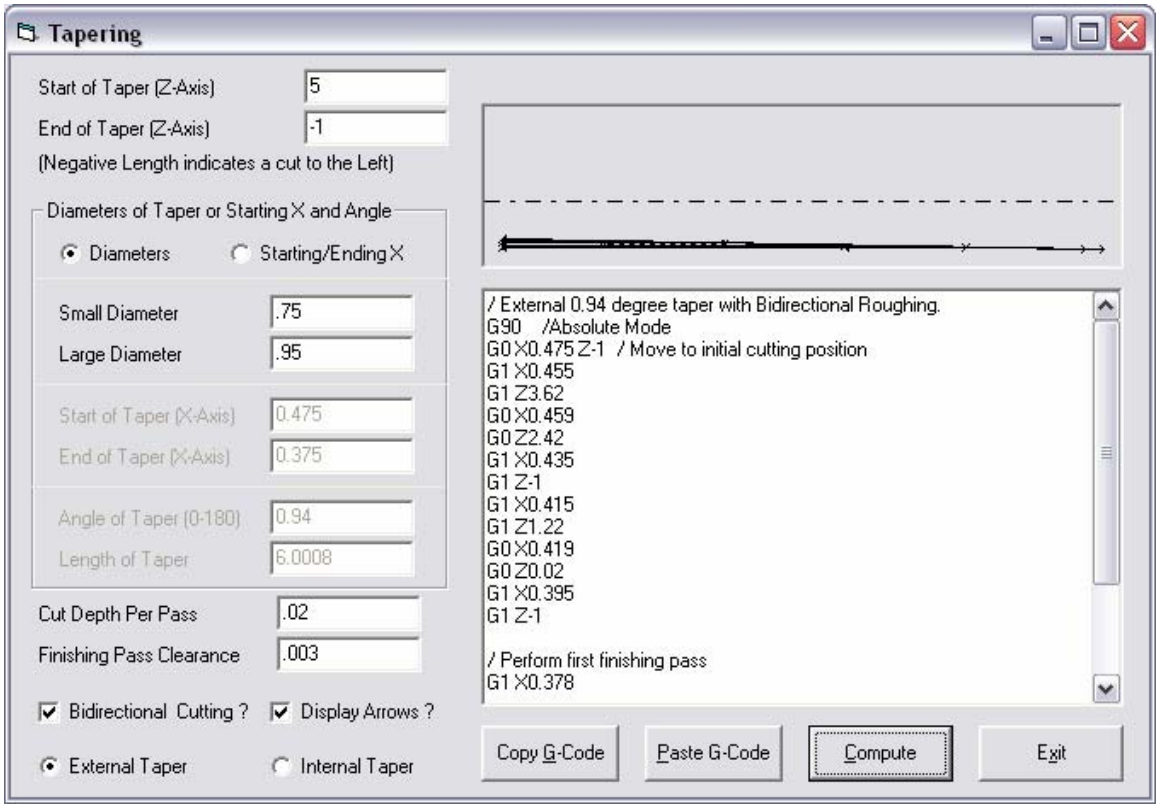
The twelfth icon in the Toolbar also corresponds to this menu command.

****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.



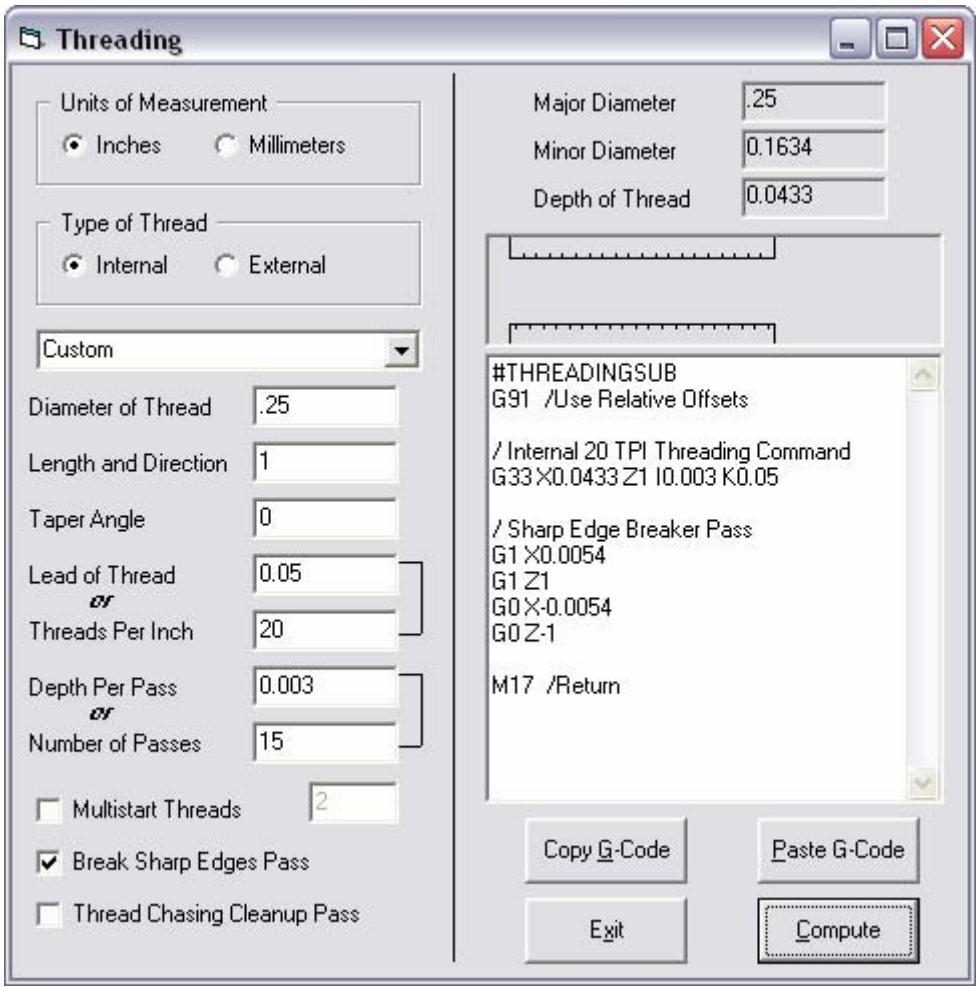
Tools- *Tapering*

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.



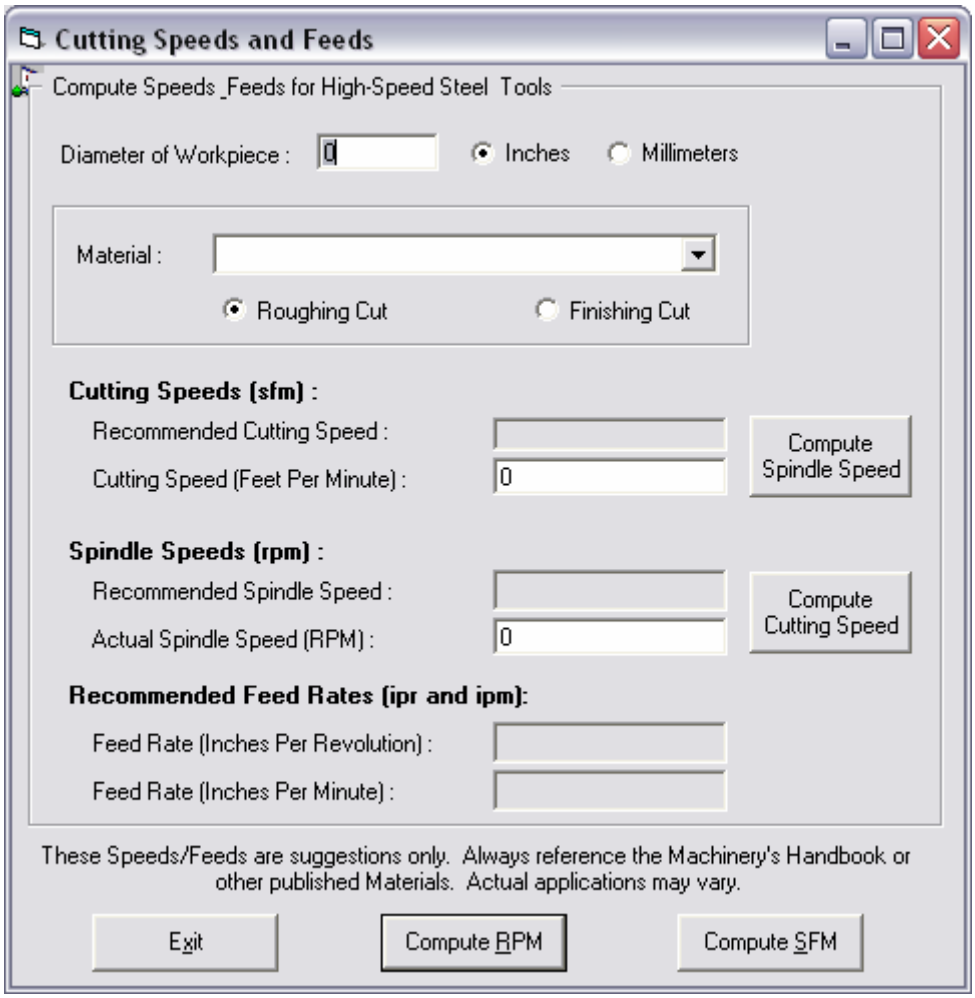
Tools- *Threading*

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.



Tools- *Speeds and Feeds*

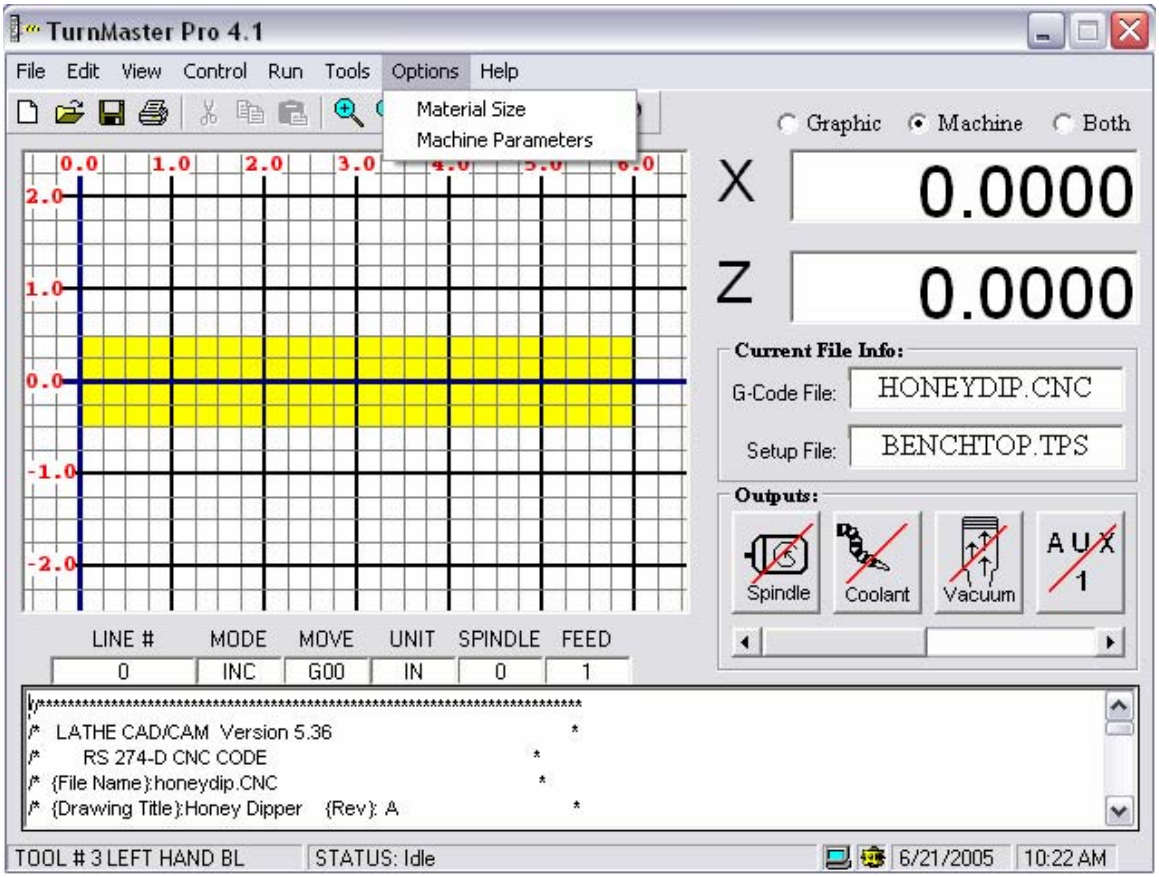
Opens the Cutting Speeds and Feeds Window which allows the user to calculate the spindle speed that they need to be running their machine at depending on material, number of teeth, and other variables which come into play when determining cutting speed. Simply input the proper data and let the program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



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

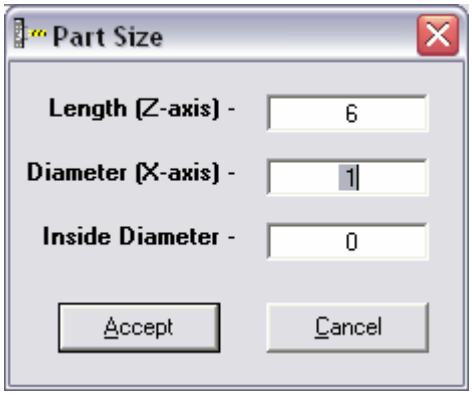
The underlined letters are called ***QuickSelect*** keys.

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



Options- *Material Size*

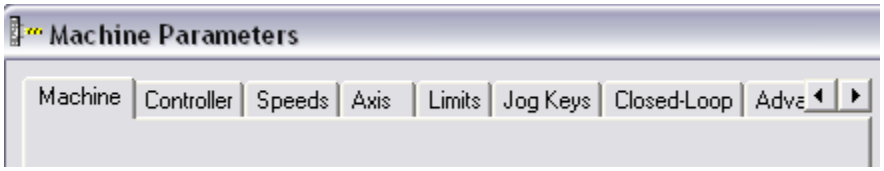
Allows the user to enter the material **Length**, **Diameter,** and/or **Inside Diameter** in the units specified in the **Machine Parameters window**.

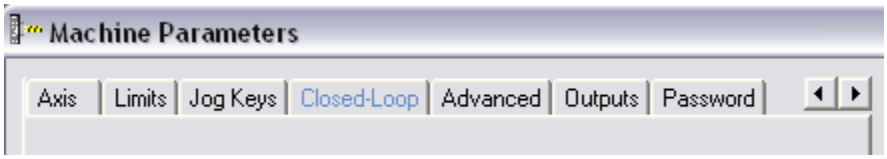


Options- *Machine Parameters*

Allows the user to set Machine Parameters such as Unit of Measurement, Origin, select the type of **MicroKinetics** controller, **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

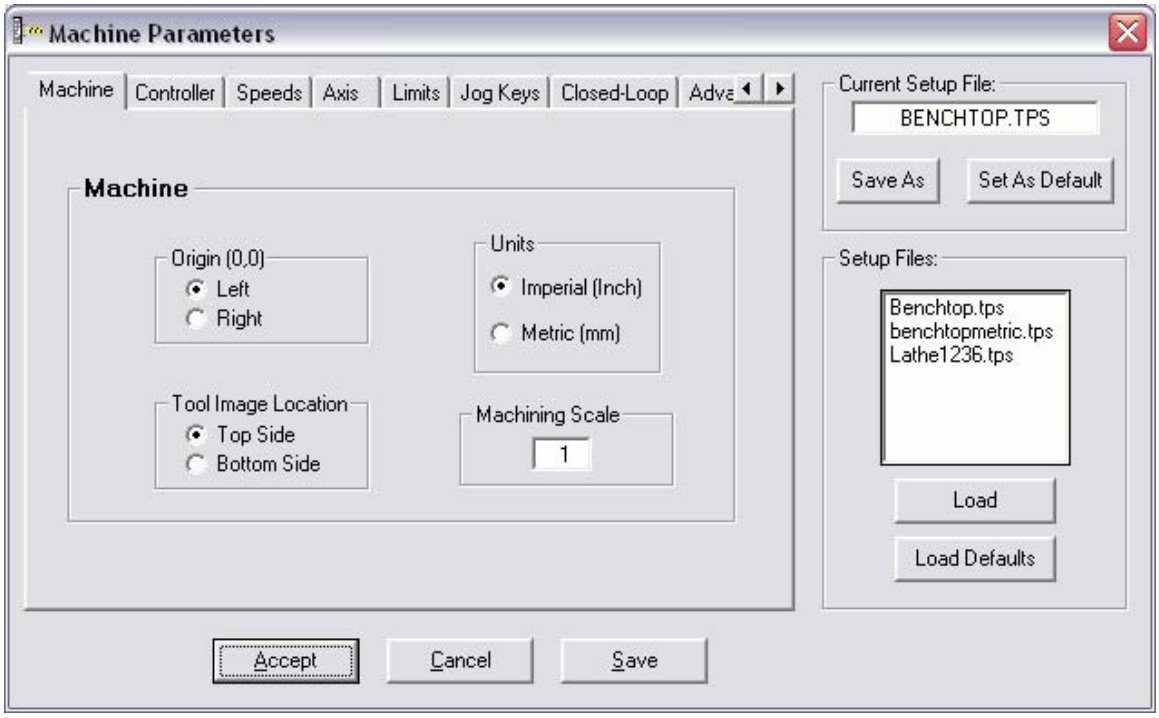
change all of these parameters. The user may also select a Machine Type. This specifies what kind of machine you will be using **TurnMaster Pro** with. This menu can be accessed with a **Mouse QuickSelect** click by left clicking in the whitespace of the "Setup File" box in the Current File Info. frame. Below are two images representing the file tabs on the machine parameters page.





Options– *Machine Parameters-Machine*

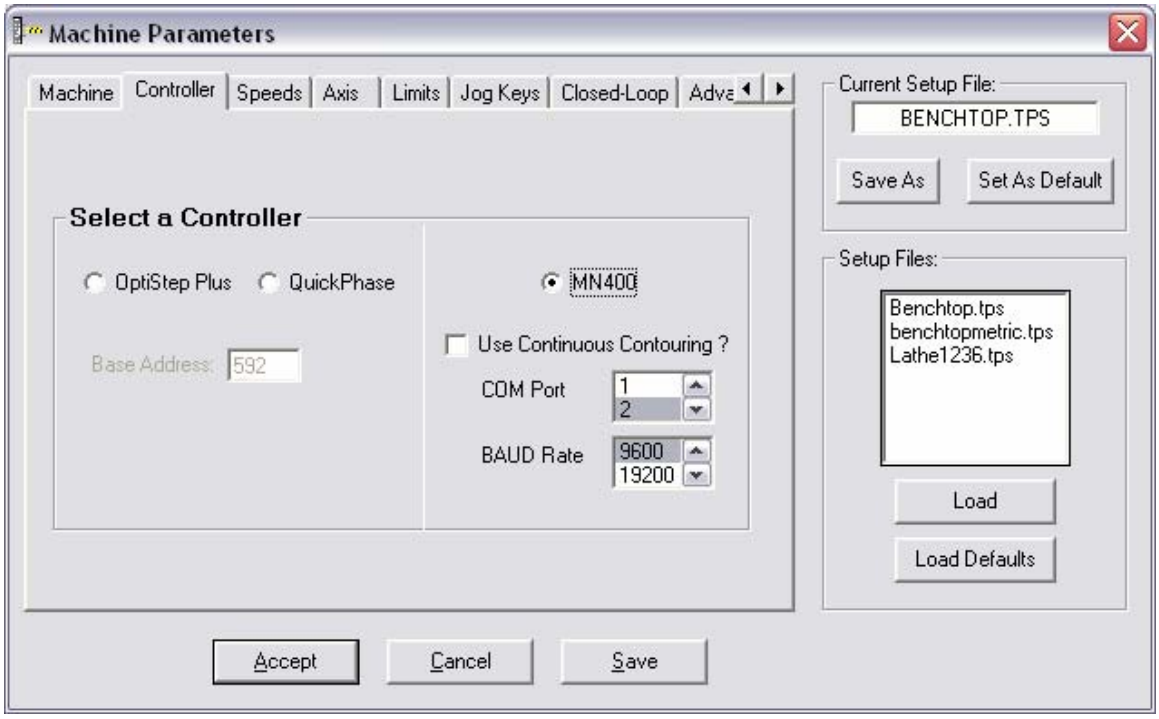
Allows the user to set the origin of the machine, unit of measurement and machine type.



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 port and BAUD Rate of their serial connection for configuration of the MN400. You can also turn on and off the Continuous Contouring feature found on the MN400 control card.

****NOTE:** The DIP switch settings on the card must match this address



Options– *Machine Parameters-Position and Feeds 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.

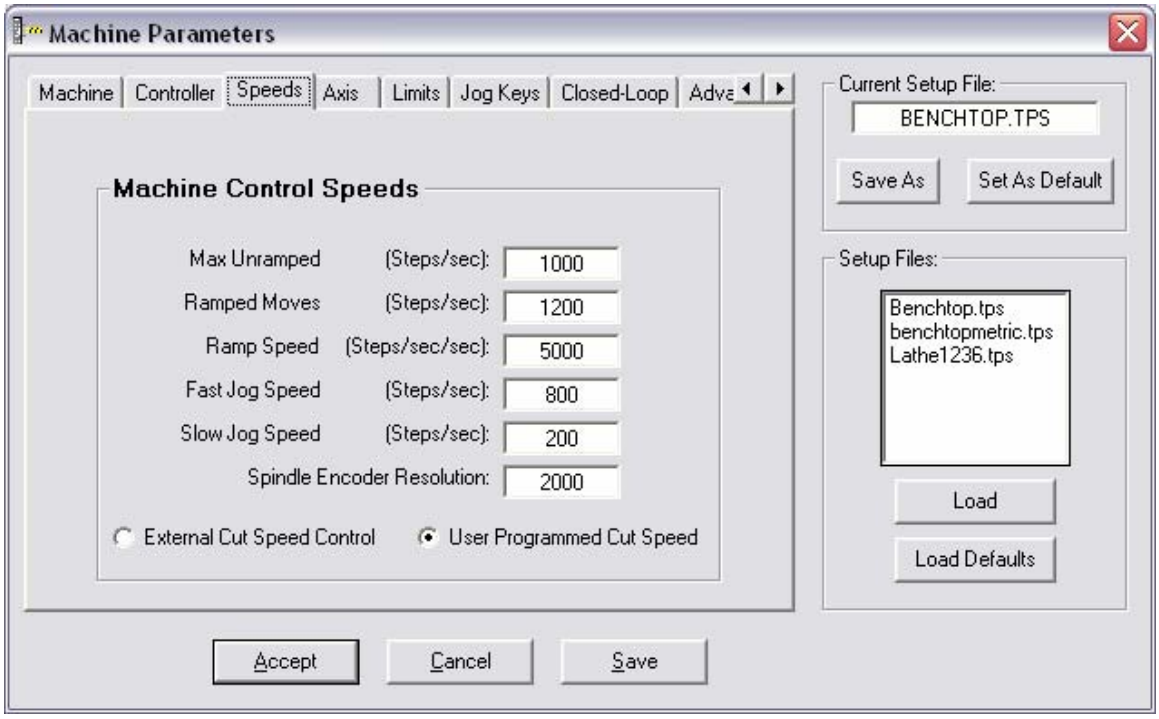
Fast Jog Speed is the rate the machine will travel in the *Jog Mode* after the "Fast Jog" button is pressed on the screen.

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.

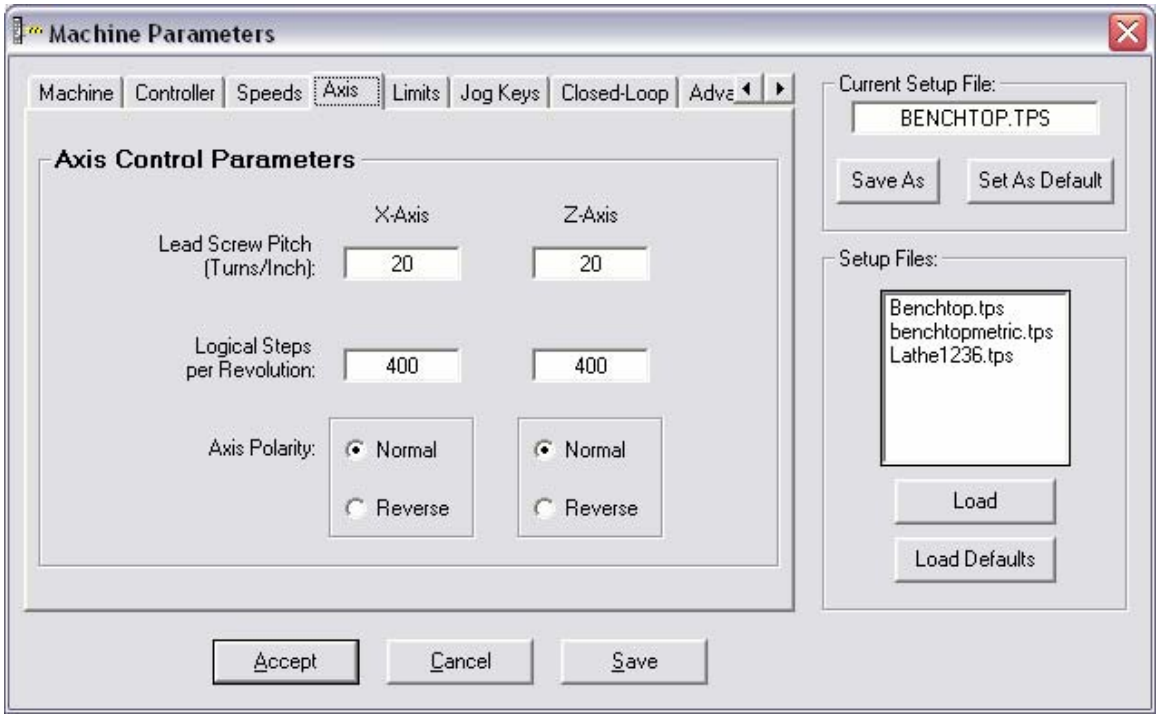
Note: If REVERSE is selected for any axis, then the limit sensor connections must also be reversed **

(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.

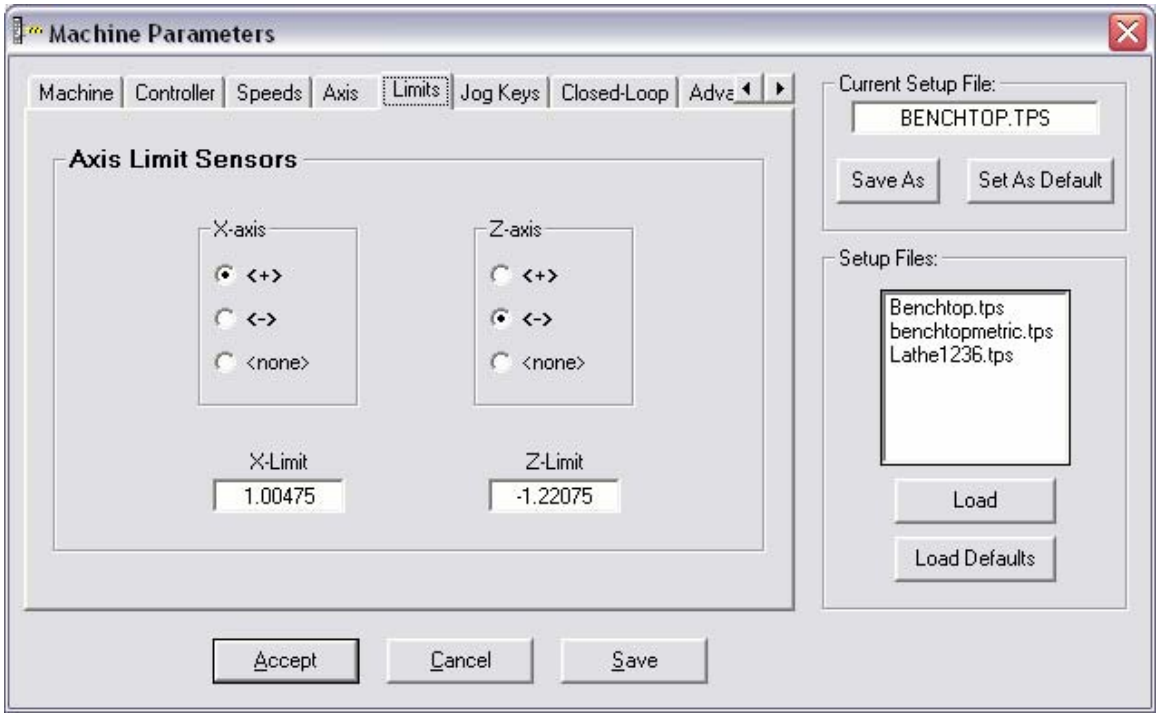
Example:

Motor Res.Driver Res.Logical Steps/Rev
200 FULLSTEP 200
200 HALFSTEP 400



Options- *Machine Parameters-Limit Sensors*

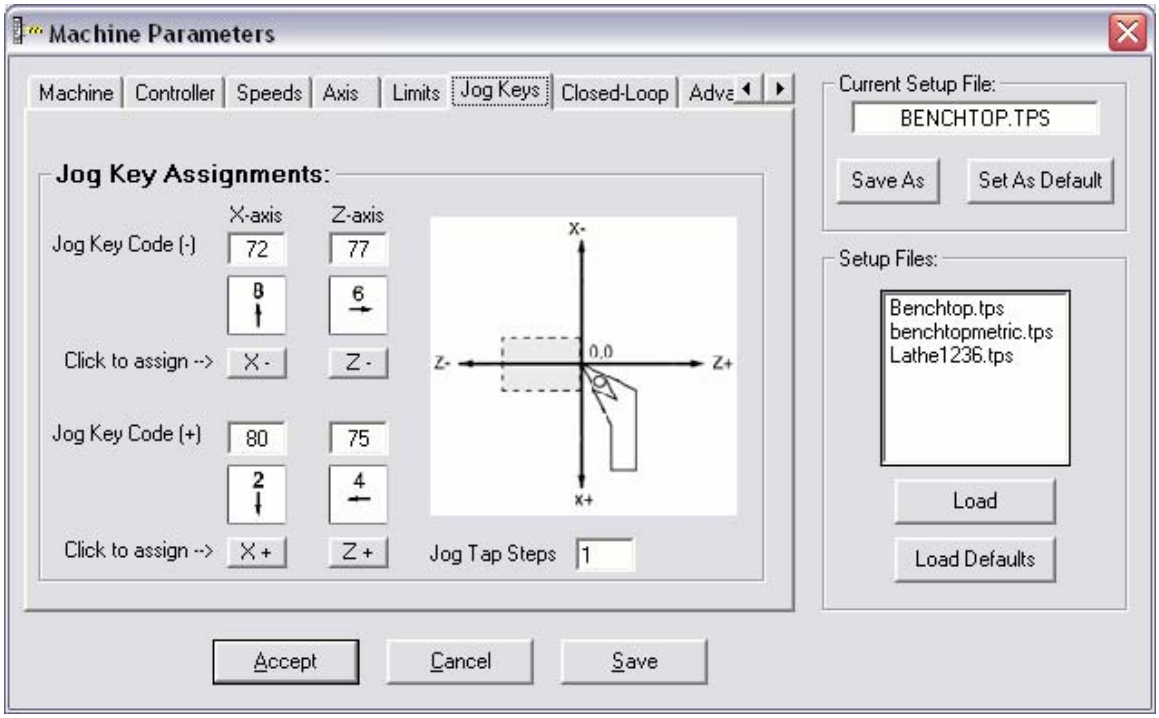
Allows the user to select the limit switches that will be used to home to. Enter the position of the home limit sensor for each axis in reference to the origin of the part. On the Z-axis only, enter the position of the bottom of the tool holder in reference to the origin of the part.



Options- *Machine Parameters-Jog Keys Assignment*

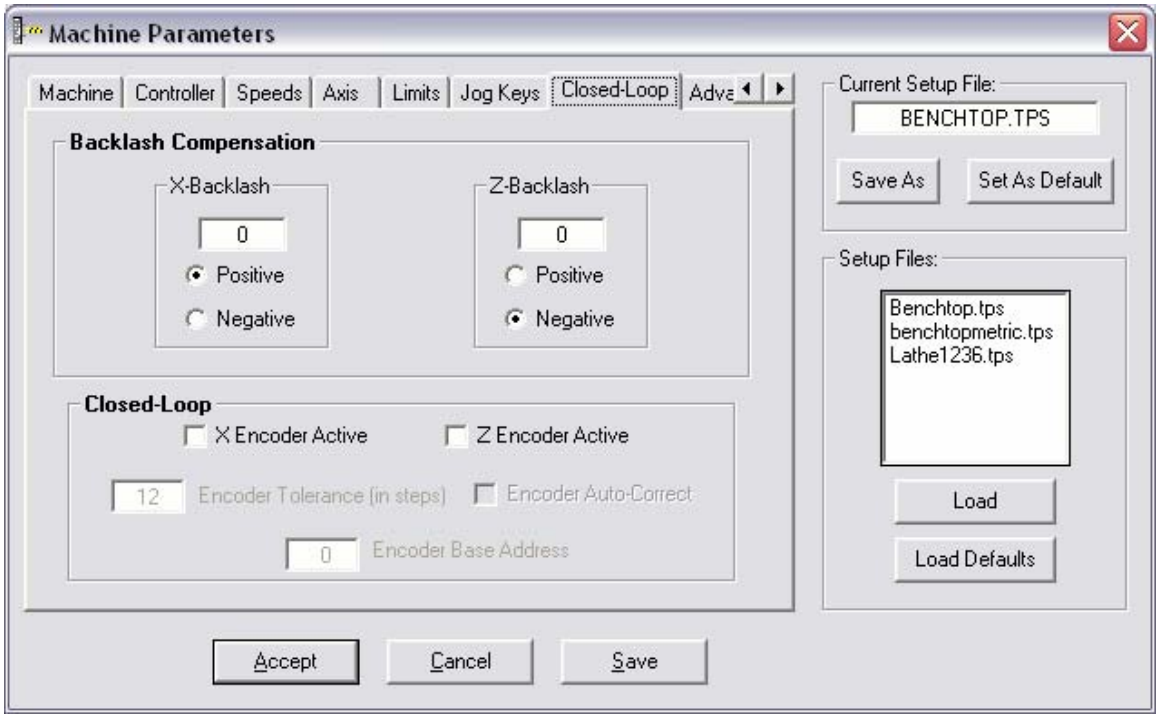
Allows the user to choose which keys on the computer keyboard control the jog movement of the machine. The default sets the Z-axis to jog with the left and right arrow keys, and the X-axis to jog with the up and down arrow keys. If you wish to change the jog keys, you should click on the button that 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-Closed Loop / Backlash compensation*

Allows the user to enter values for XYZ backlash compensation and closed loop encoder values.



Options– *Machine Parameters-Advanced*

Allows the user to enter the third and fourth axis type, if applicable, and set other attributes associated with these axes.

If you select the Tool Turret option, then you will need to enter the number of tools per turret as well as the steps per tool station.

If you select the **Spindle** option, 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.

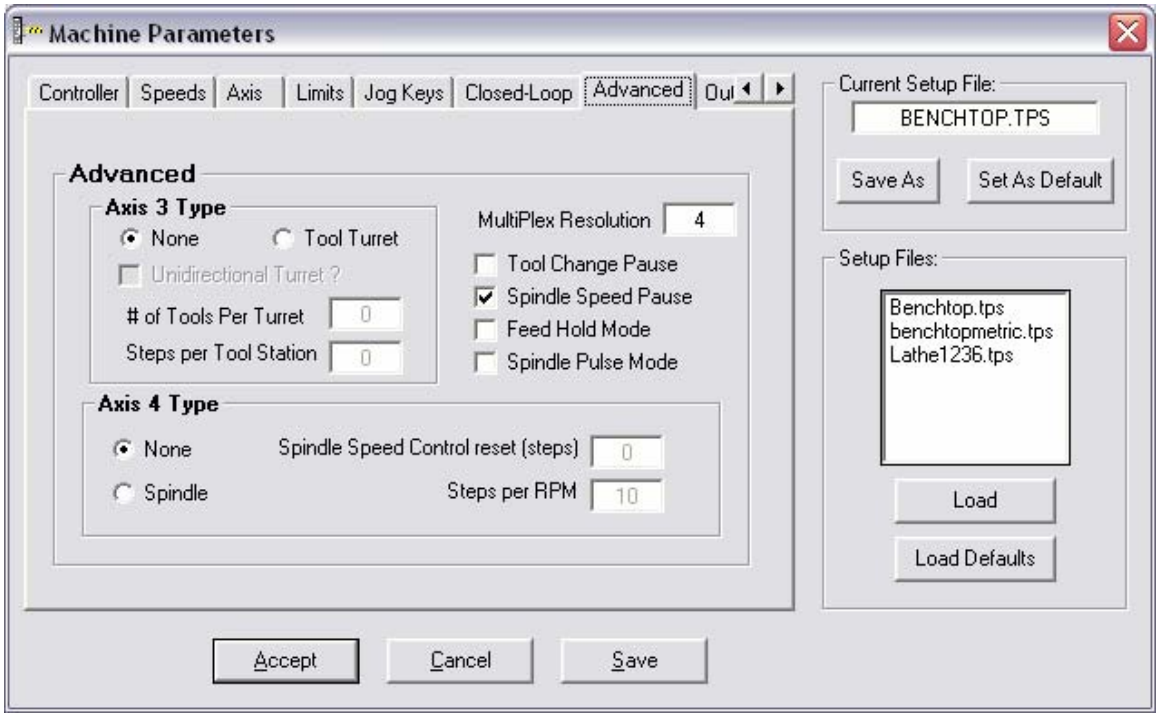
Feed Hold Mode - This parameter specifies the action TurnMaster Pro will take when the shield is opened. If checked, the software will simply pause the machining (spindle motor remains on). If unchecked, the software will stop machining, turn the spindle motor off and prompt the user to Abort or Continue.

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

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.

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.



Options– *Machine Parameters-Outputs*

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

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-Set As Default*

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

Options- *Machine Parameters-Load Defaults*

Clears the current settings and restores the defaults from disk.

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-Save Setup*

Saves the current setup parameters to the current setup file.

Options- *Machine Parameters-Save As*

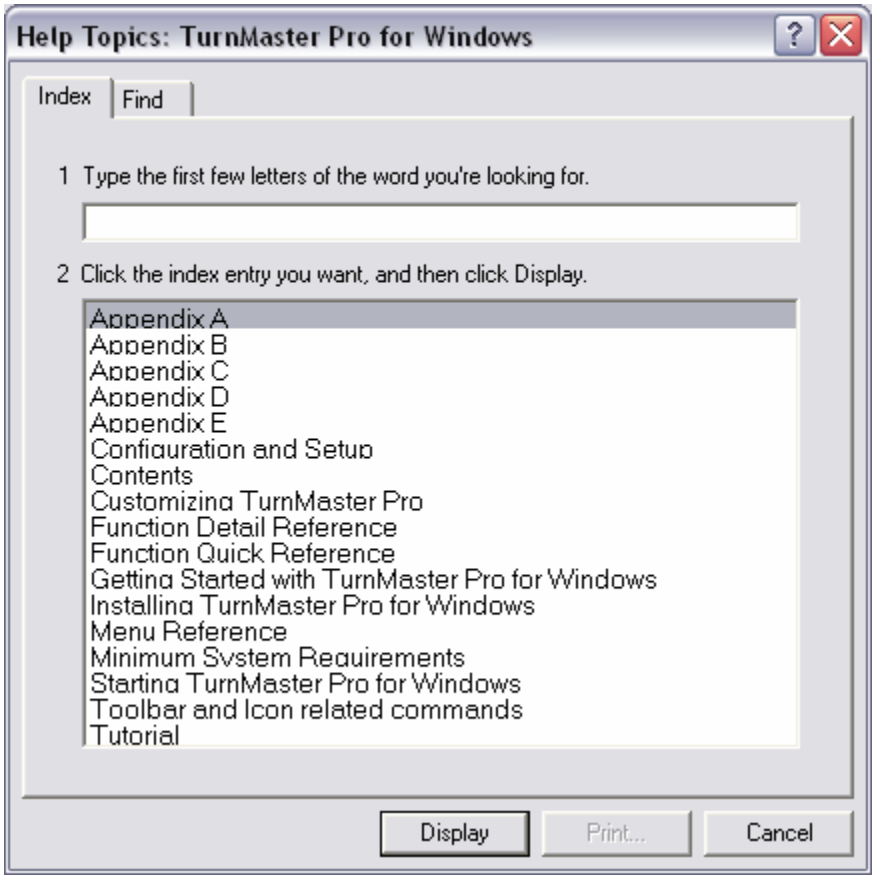
Allows the user to save the current setup parameters to a setup file of choice or create a new one.

Help- *Contents*

This will display a window containing the contents of this help file. The fourteenth icon in the Toolbar also corresponds to this menu command

Help- *Search For Help On*

This will search the help file and display a window containing the “searched for” function and a description on how to use it.



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 Library, Company, and Copyright information.


Application Information
MICROKINETICS
Corporation
TurnMaster Pro for Windows
Version 4.1.09
Instep Library v0
Turning Machine Control and CNC Program Verification
for Microsoft Windows (tm) Based PC's
(C) Copyright 2001 - Present, MicroKinetics Corp.,
Kennesaw, GA 30144
E-mail: mk-tech@microkinetics.com - Phone: (770) 422-7845
Warning: This program is protected by U.S. and International
copyright laws. Any duplication is prohibited and will be prosecuted to
the full extent of the law.
OK

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.

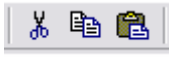
The following are short descriptions of the above graphical commands contained on the toolbar. (from left to right)

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%

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 G03command, 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.

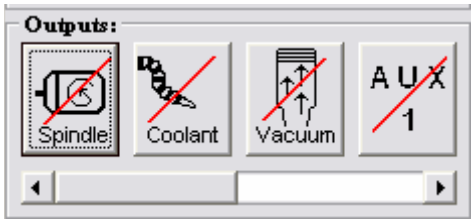
Help - Displays a window showing the contents of this help file.

ICON RELATED COMMANDS

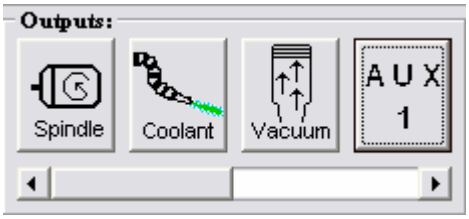
Along with the Toolbar, there are other graphical/icon related command keys.

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

OFF Position



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)**

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

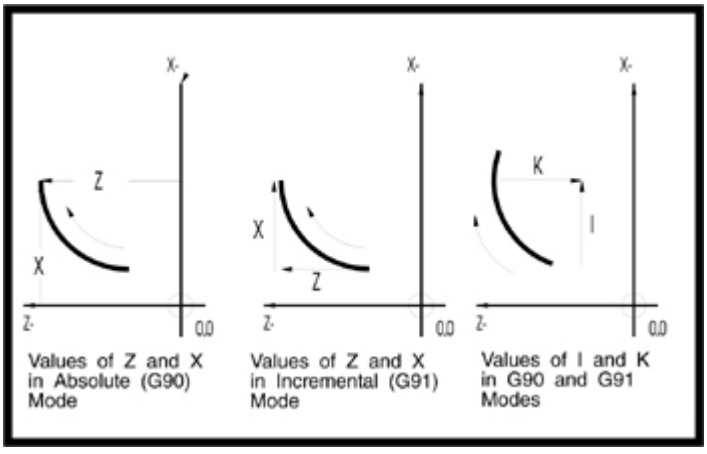
G01
This code is used to cut along the path of a straight line

EXAMPLE: G01 X1.2 Z0.1 F20

G02
This is used for a **two**-**axis** circular cutting move in a **clockwise** direction.

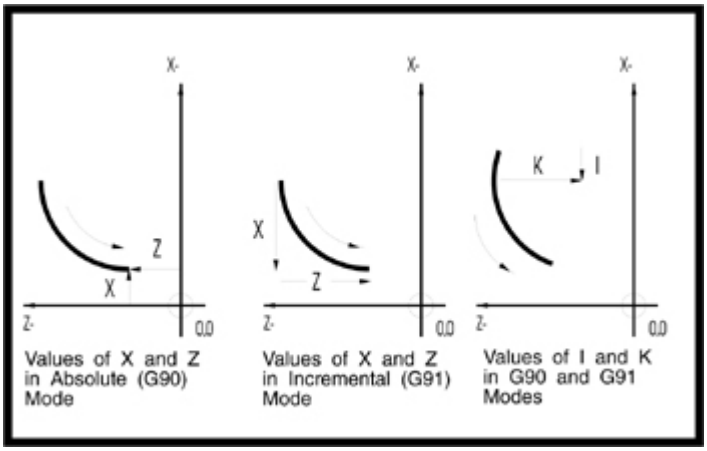
To perform a circular move the cutting tool is moved to the **starting position** before the **G02 code** is used. Following the **G02 code**, the end point of the arc is expressed in **two axes** (**X and Z**). Next, the distance from the **circle center point** to the **start of the arc** is given 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**.

EXAMPLE: G02 X.5 Z2 K1 F10



G03
This code is used for **counterclockwise** cutting moves.
It is used in the same way as the **G02 command** described above.

EXAMPLE: G00 X3 Z-.2
G03 X4 Z2.2 I.4375 F10



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 **/this pauses 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.

EXAMPLE: G25 #GROOVE

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

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 **logical zero** or if the **parameter is true**.

Note: Can be used with **Optistep-Plus** controller only.

EXAMPLE 1: G26 I1 #HOLESET

Will jump to **#HOLESET** when **input 1** is a **logical zero**.

EXAMPLE 2:

```
G28 (parts) = 5
G90          /ESTABLISH ABSOLUT
G75
G92 X0 Y0 Z0.5   /ESTABLISH START POINT
#loop
/program goes here
/
G28 (parts) = [(parts) - 1]
G26 (parts) #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.

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**

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**).

X should be 0.050" above material before command.

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

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

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 programming mode (default)**. **(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)**

EXAMPLE: G75

G80
Cancels a **CANNED CYCLE**. Use this command after each **CANNED CYCLE function**.

G81
This is a canned outside and inside diameter turning cycle. The parameters include **Z** length, **X** depth and **Q** # of passes of **I**, depth per pass.

X-axis must be 0.050" above the material before command.

EXAMPLE:

```
G00 Z.050 /Go to 0.D.+.050 & right +.050
G81 Z-.5  X.2  Q8 F20
```

Will turn down the material to 0.60" diameter in 8 passes.

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 **I**, 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

Will face 0.010" off the stock in 3 passes. The **X** value indicates that 0.500" diameter stock is in the machine.

G90

This code Sets the **absolute programming mode**. In the **absolute 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. The **default programming mode is incremental**.

EXAMPLE: G90

G91

This code sets the **incremental programming mode**. In the **incremental 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 programming mode** is **incremental**.

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.

Miscellaneous Functions (M-Codes):

M00

This command causes a **temporary stop**. Operation **resumes** by pressing **<ENTER>**

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)**

M04
This turns **<ON>** the spindle motor **counterclockwise (control output #5)**

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**

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 homes the **X axis** and then the **Z axis**.

M39
This **<CLOSES>** the chuck **(control output #6)**

M40
This **<OPENS>** the chuck **(control output #6)**

M66
This allows for jogging of the axis 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 and position will be restored to the exact location they were before the M66 command was encountered.

M97
This command pauses **part program execution** until a **logical zero** is detected at the specified input.

Note: Can be used with an **OptiStep Plus** controller only.

EXAMPLE: M97 I2

Will wait for **input 2** to become a **logical zero** 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**
G70	**Set inch programming (default)**
G71	**Set metric programming**
G74	**Sets single quadrant arc programming mode (default)**
G75	**Sets multiple quadrant arc programming mode**
G80	**Cancel canned cycle**
G81	**Canned cycle for turning down outside diameter**
G82	**Canned cycle for reducing part length (facing)**
G90	**Set absolute programming mode**
G91	**Set incremental programming mode (default)**
G92	**Set current tool position counters to value**
G95	**Chain to next part program**

Miscellaneous Functions (M-Codes)

M00	**Temporary stop**
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**
M39	**Chuck <CLOSE> (output #6)**

M40	**Chuck <OPEN> (output #6)**
M66	**Enables Jogging during a program**
M97	**Wait for true input state then continue**
M99	**Restart part program from beginning**

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***.

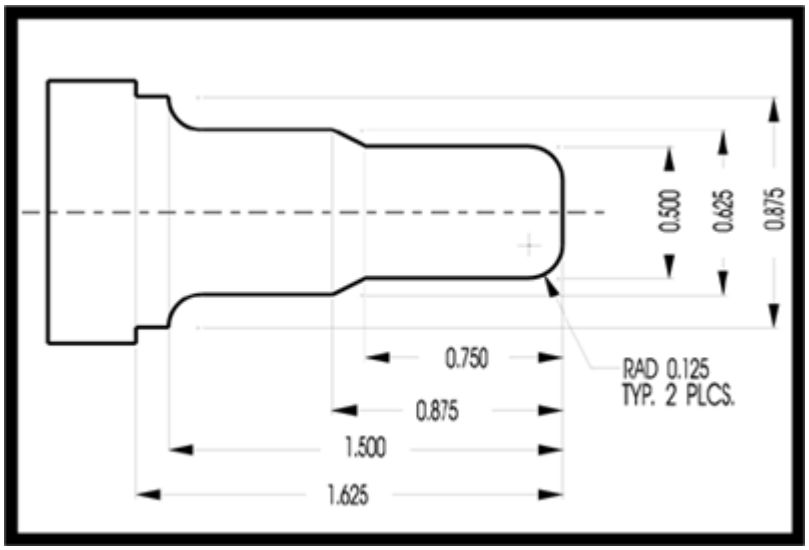


Figure 5.1.1 Part Profile Drawing

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                 Z0       /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               Z -.750             /MACHINE TO THE TAPER
G01   X .3125  Z -.875          /MACHINE TAPER
```

```
G01                    Z-1.375          /MACHINE TO SECOND RADIUS
G02   X .4375  Z-1.500  I  .125    /MACHINE SECOND RADIUS
G01                    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 for Windows. The screen shown in ***Figure 5.2.1*** should be on your desktop.

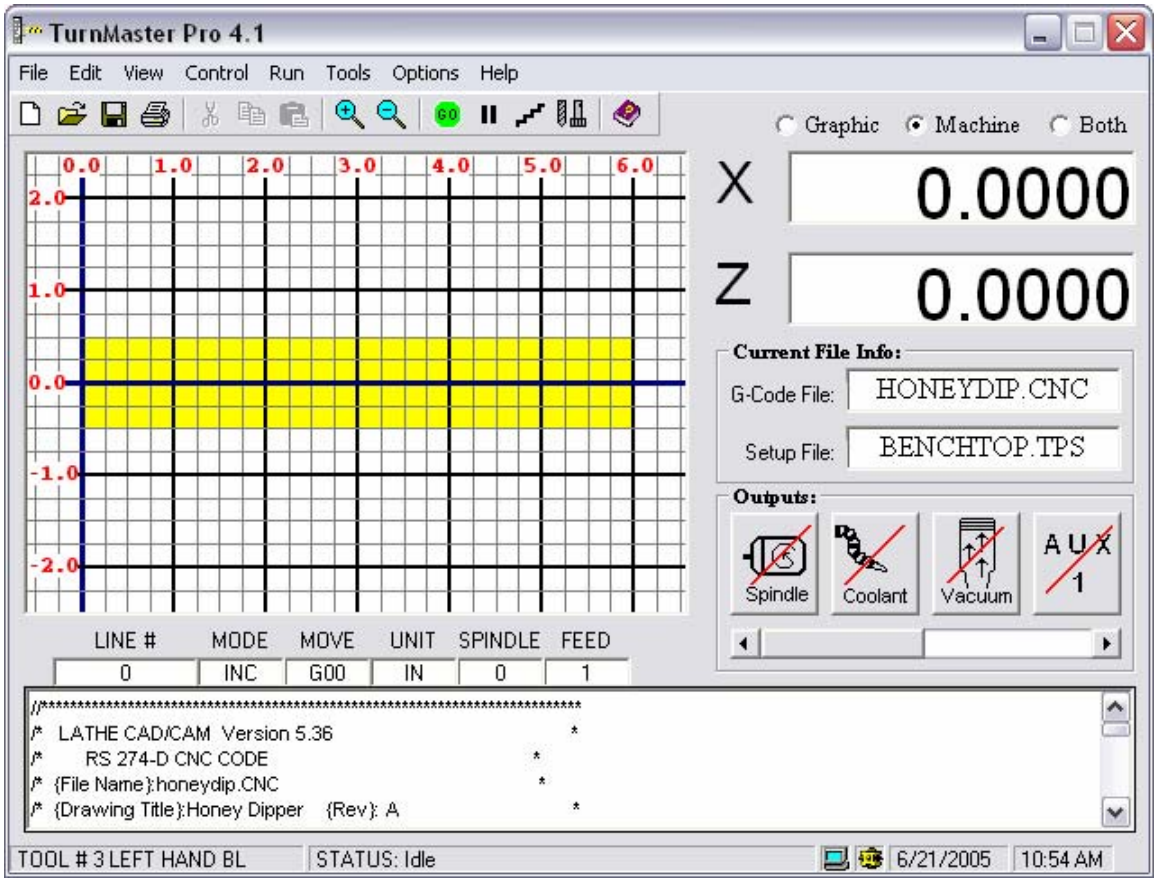


Figure 5.2.1 TurnMaster Pro Graphical Screen

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
   Chucking Method = Lathe Chuck
   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***.

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** 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***.

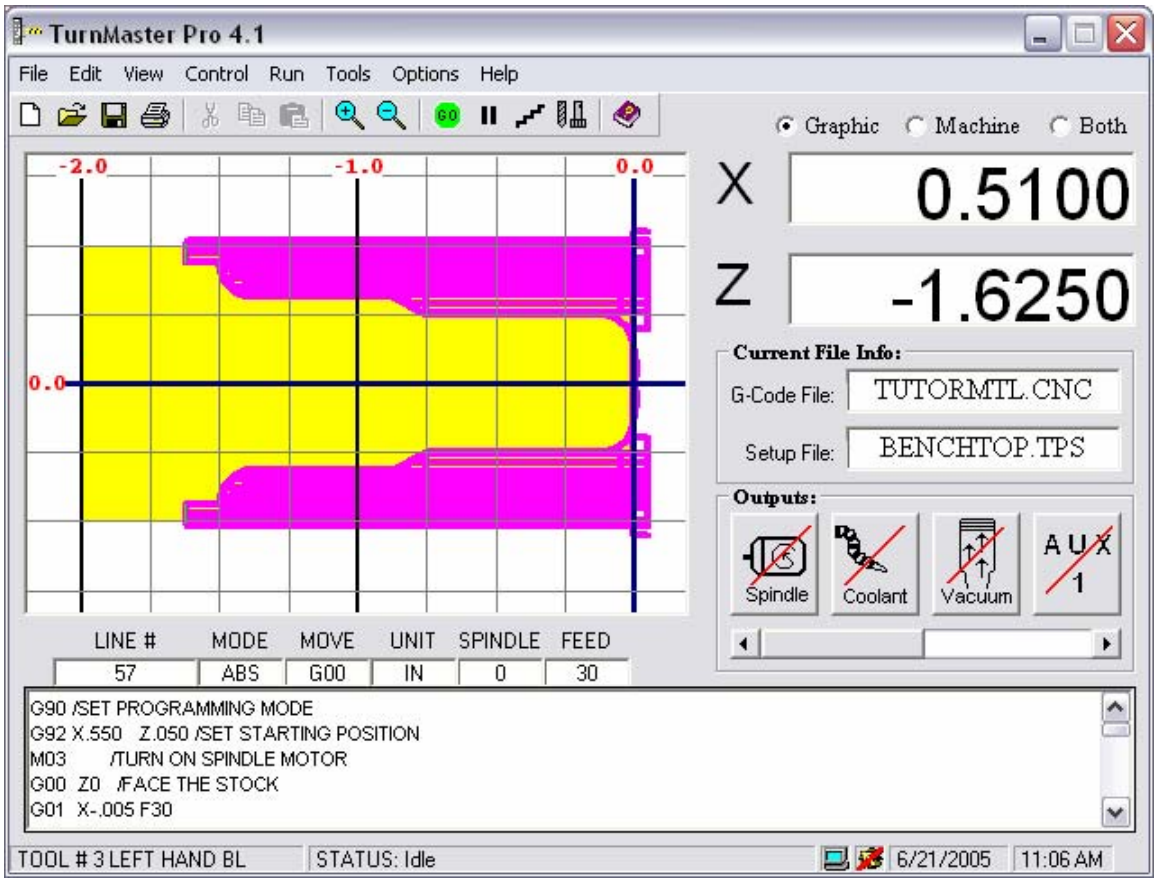


Figure 5.2.4 Tutorial CNC Simulation

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

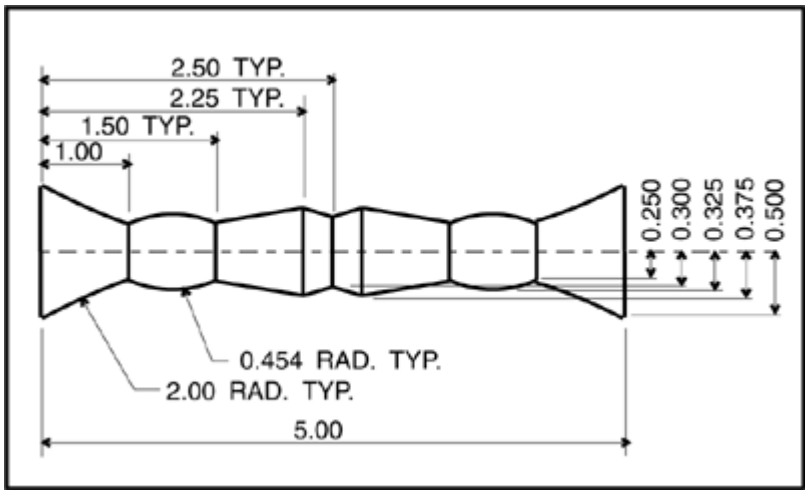
4. Click **Machine Mode** with the mouse. Note that the status line at the bottom of the screen indicates **machine mode** as well as the radio button selection in the top right corner or the window.

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

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 **chucking method** for the metal example is a **lathe chuck**, and in the wood example it is between **centers**.

3. 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**).

4. 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:

   Chucking Method = Centers
   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 or 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>*** *will pause the machining process at any time. If something starts going wrong during machining, PRESS THE EMERGENCY STOP button on your DriveRack!*

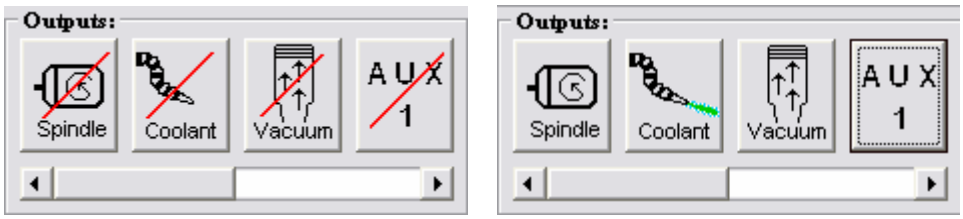
Customizing TurnMaster Pro

This section will instruct you on how to customize TurnMaster Pro for Windows.

TurnMaster Pro for Windows 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 for Windows 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.

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
F2 – 60
F3 – 61
F4 – 62
F5 – 63
F6 – 64
F7 – 65
F8 – 66
F9 – 67
F10 – 68
1 – 2
2 – 3
3 – 4
4 – 5
5 – 6
6 – 7
7 – 8
8 – 9
9 – 10

0 – 11
A – 30
B – 48
C – 46
D – 32
E – 18
F – 33
G – 34
H – 35
I – 23
J – 36
K – 37
L – 38
M – 50
N – 49
O – 24
P – 25
Q – 16
R – 19

S – 31
T – 20
U – 22
V – 47
W – 17
X – 45
Y – 21
Z – 44
- – 12
= – 13
\ – 43
[– 26
] – 27
; – 39
“ – 40
– 28
, – 51
. – 52
/ – 53

Left ALT – 56
Left SHIFT – 42
Right SHIFT – 54
Caps Lock – 58
SCR LOCK – 70
NUM LOCK – 69
DEL – 83
INS – 82
TAB – 15
CTRL – 29
ESC – 1
SpaceBar – 57
BKSPC –14
Keypad + – 78
Keypad - – 74
Keypad * – 55

Keypad 1 – 79
Keypad 2 – 80
Keypad 3 – 81
Keypad 4 – 75
Keypad 5 – 76
Keypad 6 – 77
Keypad 7 – 71
Keypad 8 – 72
Keypad 9 – 73

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

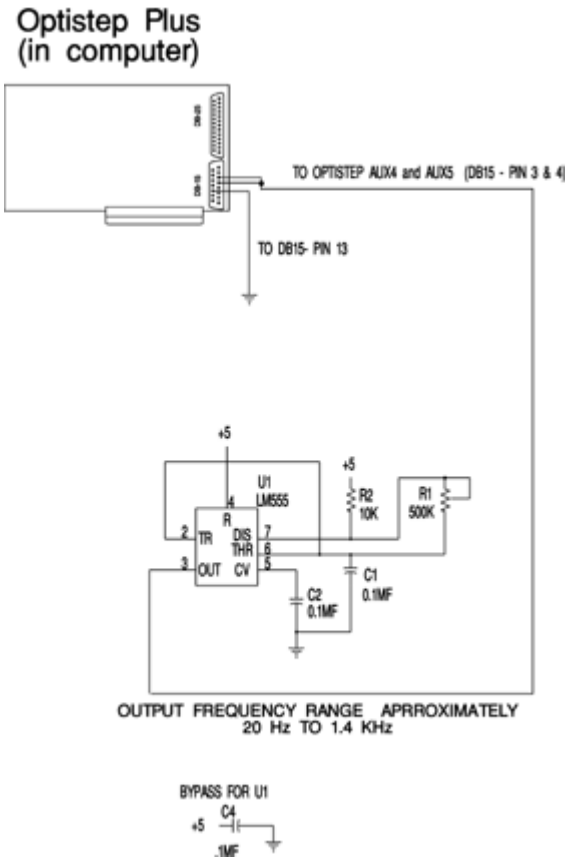
Connect an external pulse source, as in the **example below**, to the **OptiStep Plus controller** to use the **External Cutting Speed Control option**.

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.

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: 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-Axis TPI x Steps/Rev < Encoder Res. x Min. TPI to be cut

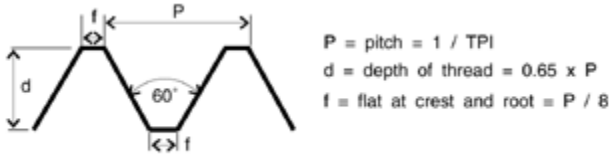
where **Z-Axis TPI** is the threads per inch of your lead screw, **Steps/Rev** is the resolution of the stepper motor driving the axis, **Min. TPI to be cut** is the minimum number of threads per inch to be machined, and **Encoder Res** is the resolution of the encoder on the spindle shaft.

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 TPI) x (400 Steps/Rev) < Encoder Res x (5 TPI)
so: Encoder Res > 1600 pulses / Rev

Select an encoder that has OPEN COLLECTOR outputs and a Vcc = +5 Vdc.

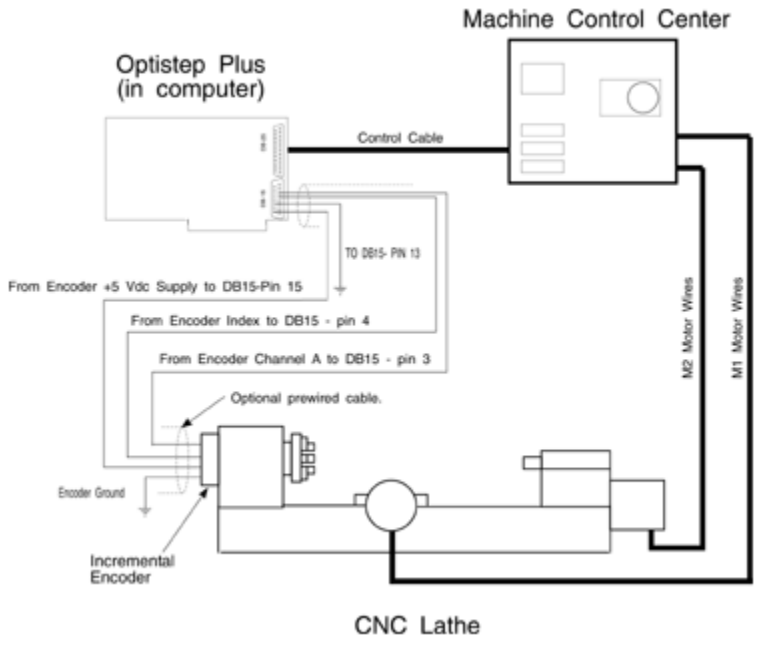


The American National form of thread

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 For Windows** 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.

These attributes are displayed, in order, below.

Version = 1.0
Card address = 592
Card type = OptiStep plus
Max depth per pass = 1
Maximum unramped speed = 62
Maximum speed = 62
Acceleration rate = 62
Fast jog speed = 1000
Slow jog speed = 500
External sync active = 0
X screw pitch = 20
X steps in rev = 400
Z screw pitch = 20
Z steps in rev = 400
X plus = 75
X minus = 77
Z minus = 32
Z plus = 22
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 = 0
Diag Speed Increment = 0
X Distance = 0
Z Distance = 0
Tolerance = 0
Default tool = 0
Xbacklash = 0
Zbacklash = 0
Xbacklash Direction = 1
Zbacklash Direction = 1
Encoder Tolerance = 6
X Encoder Active = 0
Z Encoder Active = 0
Encoder Auto Correct = 0
Jog Tap Steps = 12
Feedhold Mode = 0
Multiplex = 0
OptiTracker Address = 0
Spindle Pulse Mode = 0
Axis #3 Type =
Axis #4 Type =
Steps Per Deg RPM = 1
Home Speed Pot Steps = 0
Oem Name = John Doe
Oem Company = Doe, Inc.
License Number = 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 5 programmable Outputs.* **

Tool Number = 1
Tool Name = drill
Tool Picture Filename = c:\mmpro\pics\drill.bmp
X Tool Offset = 1.1

Z Tool Offset = 1.2
Tool Nose Radius = .125
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"**.

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

Version

This parameter specifies the current **TurnMaster Pro for Windows** 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

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:

$$\text{Steps/Sec} \setminus \frac{\text{Screw Pitch x \#Steps/Rev}}{60}$$

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)

X, Z Origin

These parameters specify the **X** & **Z** origin of the tool.

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**.

X, Z part dim

These parameters specify the **X** & **Z** dimensions 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 to 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**.

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.
.

MN400 Controller Information

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

Within this version of MillMaster 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:** After connecting the MN400 to the serial port on the back of your machine and choosing MN400 in the Machine Parameters, you are unable to jog, switch the cable to the other available serial port and re-try.

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

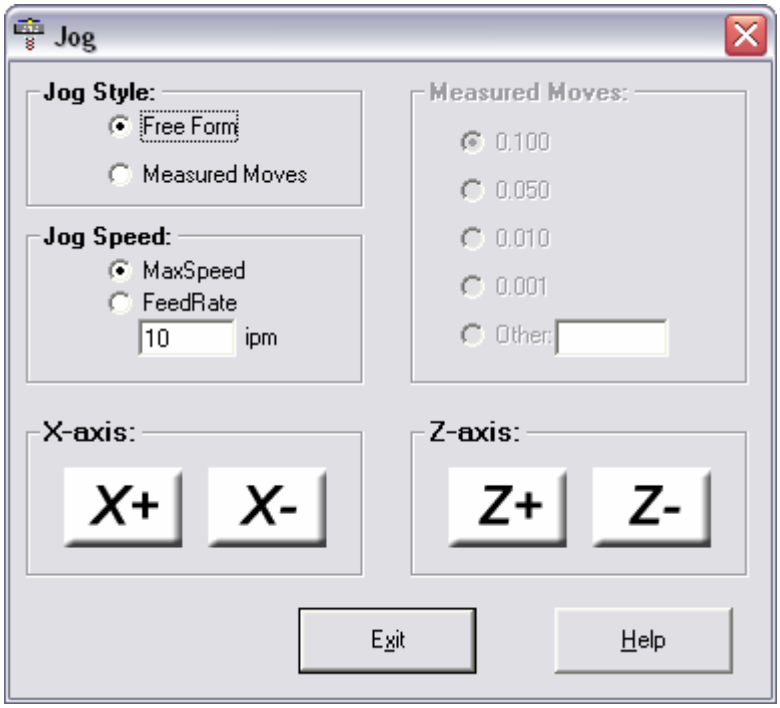
Although the operation of the MN400 is extremely close to the operation of the Instep Library, the MN400 does have some small differences. When running a program with the MN400, you do not experience the "Lockout" that occurs when using Instep and either the OptiStep or QuickPhase cards. The MN400 is 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

When the MN400 motion controller board is selected for use with MillMaster Pro, the jogging feature within the MN400 must be turned ON and OFF to operate correctly. When you choose Jog Tool from the Control menu and MN400 is selected, the software will send the command to set Jog mode.

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 button in the bottom right corner with that say 'Help' on it. This will display a message window giving you basic instructions for jogging with the MN400 controller.