Machinist ToolBox® v7.1.3

Distributed by Bair Technologies
E-Mail: sales@ebair.com
WWW: http://www.ebair.com

Machinist ToolBox® v7.1.3 is the latest version of the popular shop utility software.
It is designed with the MACHINIST in mind, but Designers, Engineers, Programmers,

Operators and Students will also benefit from it's many useful functions.
It is the result of years of real world shop-floor experience.
That experience means greater productivity because, Productivity Doesn't Just Happen!

Machinist ToolBox® v7.1.3 is the next generation of the program.
Version 7.x has been completely rewritten from scratch to take advantage of many undocumented Windows® API functions. All of the functionality is now contained in a single ActiveX *.DLL. This has resulted in a program that is significantly faster and manages memory more efficiently all while having increased functionality.

We now offer this ActiveX *.DLL for sale as a plug-in option for editNC™ v8.3.
The Machinist ToolBox® Plug-In allows users to take advantage of the inherent power of Machinist ToolBox®, directly from a menu item in editNC™.

Major Feature Updates:

***International Language Support**
-All Menus, label, captions and other dialogue text are populated from a text file.
These various language files are available directly from CNC Machinist Software.
-Machinist ToolBox® ActiveX supports the use of a comma as a decimal separator.

***Machinist ToolBox® is 100% Windows® compliant!**
-Installation uses latest Windows® Installer Technology
-Machinist ToolBox® FULLY supports WindowsXP.
-Modified User Interface

***Main Dialogue**
-Machinist ToolBox® stores last main window size and position on close.
Default size and position can be reset.

***Function Dialogues**
-ALL function dialogues stores last screen position on close.
Default position for all dialogues can be reset.

***Conversion Functions**
-NC File IJ/IK Arc to R-Word conversion now supports incremental
and absolute arc centers.
-Polar->XY/XY->Polar Updated
Bug fix for bug that incorrectly output the supplemental angle
when calculating XY->Polar.

***Speed/Feed Functions**
-Default material surface-speed unit selection added.
Users can now work in either Inch or Metric
-Milling Chip load may be either inch or metric regardless of default unit.
-Turning depth-of-cut may be either inch or metric regardless of default unit.
-Fine-Tuned UPR calculation

***Trigonometric Functions**
-True Position Tolerance calculation added.
-3-Point Arc Center calculation updated to include calculation of radius.
-Bi-Directional Polar Dimensioning (Polar->XY/XY->Polar) added.

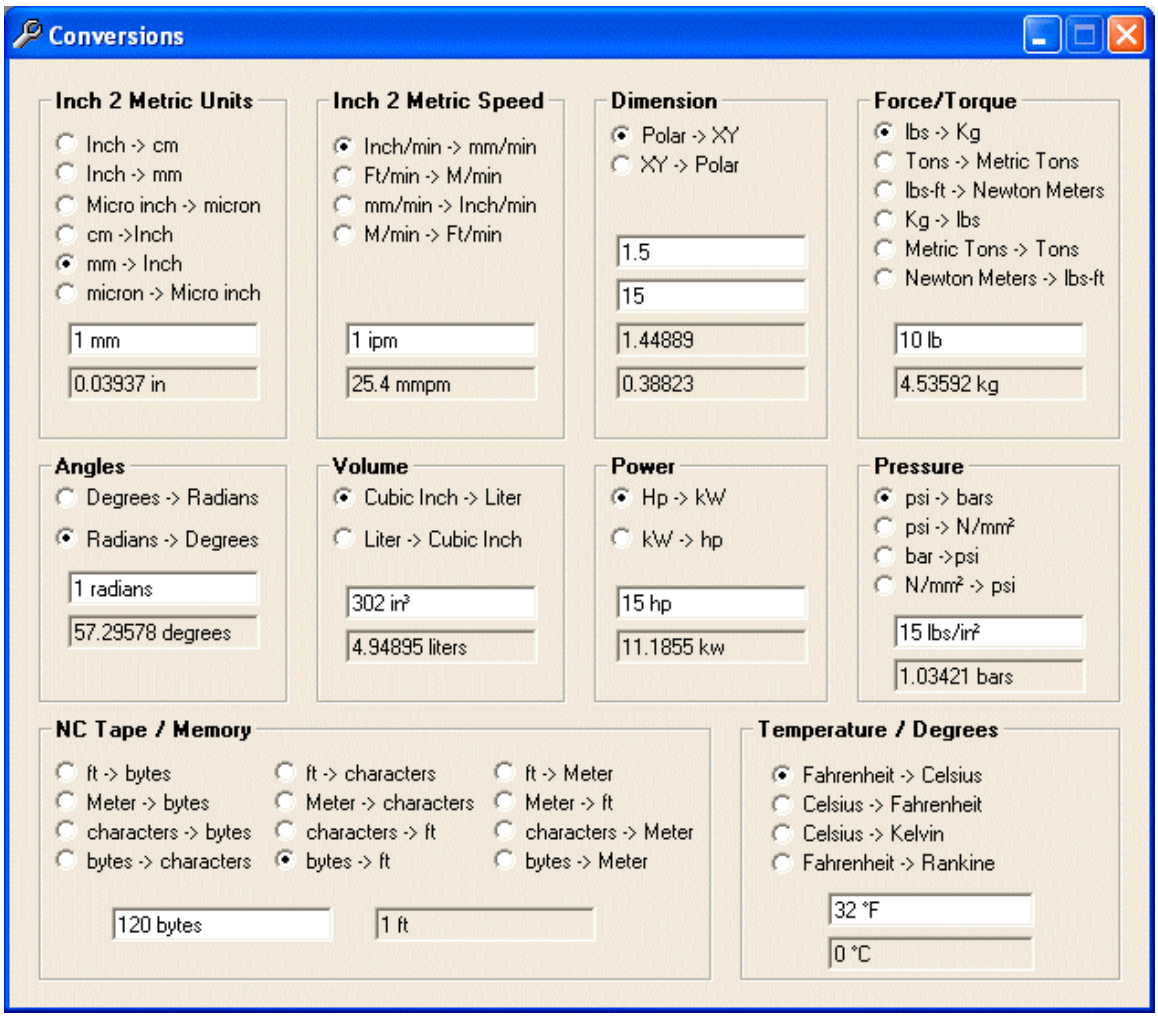
-Universal Angle Head Updated
Bug fix for angular position with an angle of A=90 or B=90 that prevented
full linearization of the cycle.

***Editor Functions**
-Direct launch of editNC® from Machinist ToolBox® [Editor] menu item.
On first launch of Machinist ToolBox®, if editNC® is detected on the
local machine, the [editNC®] menu item is enabled.

***Tip of the Day**
-Machinist ToolBox® many new tips have been added.

***Help File**
-The Machinist ToolBox® Help file has been updated to reflect all the changes

Conversions



The conversions page in Machinist ToolBox® offers a wide variety of useful bi-directional unit conversions.

Simply select the desired conversion option, key in the value you wish to convert and press enter. The converted value will be displayed in the grayed field along with the appropriate units. Clicking your mouse in any of the white input fields will automatically clear the field so you may enter another value for conversion.

The following conversions are available.

Inch 2 Metric Units
These will convert American Inch units to International Metric units and vice-versa. This is extremely useful for those who may not work with either INCH or Metric data on a regular basis.

Inch 2 Metric Speed
These will convert American Inch speed units to International Metric speed units. This is quite useful for quickly converting the machining surface speed units and machining feed rate units.

Dimension
This will covert Polar Angular data to Cartesian coordinate data and vice-versa. The Cartesian Coordinates X0, Y0 are assumed as the default center point for the polar calculation and as the reference point for the Cartesian coordinate calculations.

Conversions

Force/Torque
These will convert American Inch force and torque units to International Metric units and vice-versa.

Angles
This will convert degrees to radians and vice-versa.

Volume
This will convert Cubic Inch units to Liters and vice-versa.

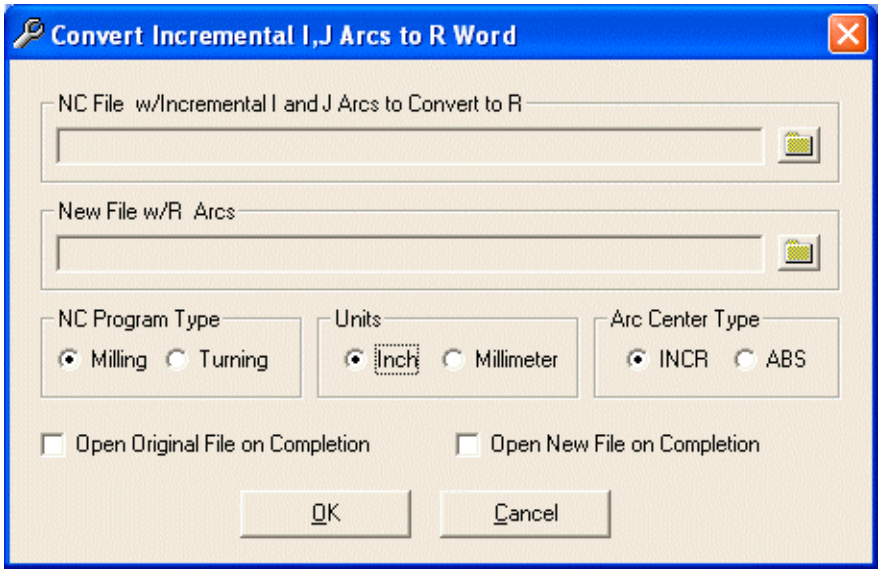
Power
This will convert Horsepower units to Kilowatts and vice-versa.

Pressure
This will convert American Inch pressure units to International Metric units and vice-versa.

NC Tape/Memory
This is one of the most useful conversion functions available ANYWHERE! Many CNC Machine Tool companies still use either characters or length of NC tape as the default unit for control memory. With these you are able to quickly convert character or tape length to computer memory bytes and vice-versa.

Temperature
These will convert the appropriate temperature degree units between the Fahrenheit, Celsius, Kelvin and Rankine temperature scales.

Convert IJ/IK Arcs to R-Word



This conversion tool allows the user to convert an NC program that uses I, J and I, K values to describe the center of an arc to an NC program that uses R-Words to describe the radius of the arc.

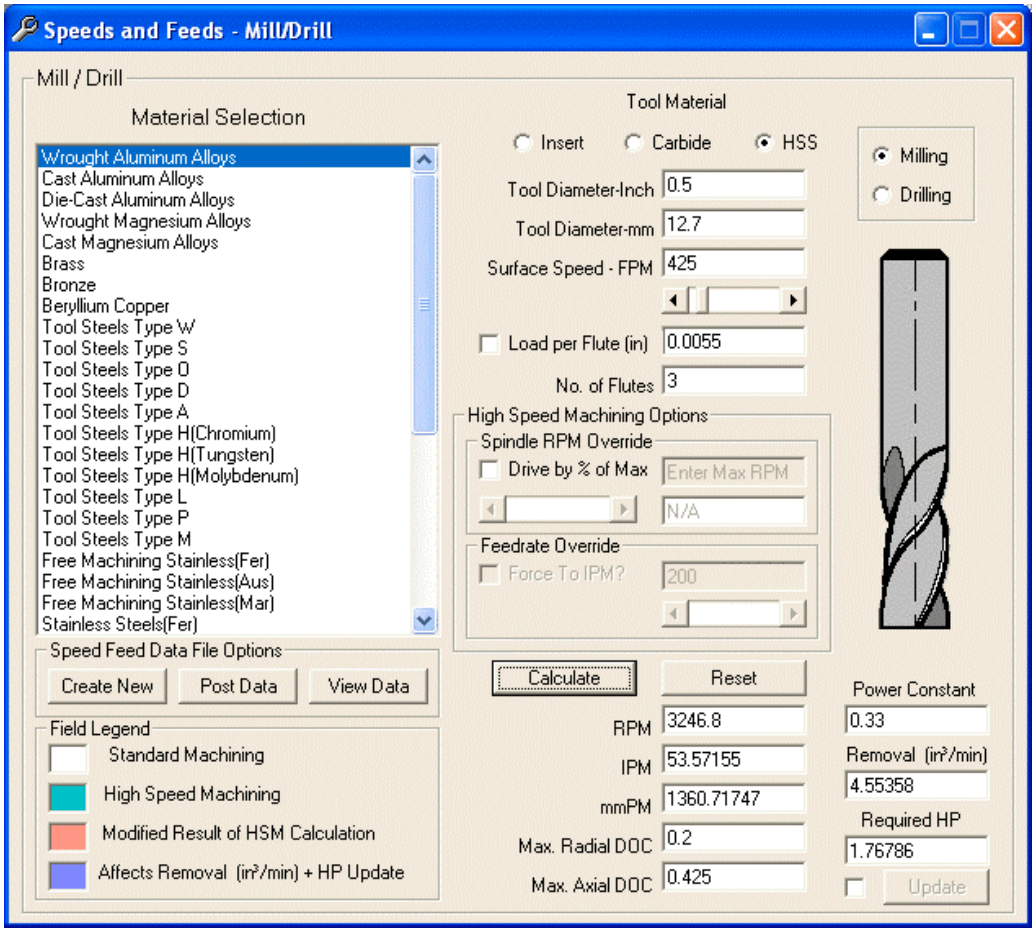
This function supports the conversion of both Incremental IJ/IK arc centers as well as Absolute IJ/IK arc centers

Simply select the original file to convert, the new file name to save as, the program type, the units of measure and the arc center type. The NC Files should be formatted per Fanuc™ NC file standards with a % character at the beginning and end of the NC file and an O-Word Program Identifier.

The user also has the option to open both the original and converted files upon completion of processing.

Click OK when ready to process.

Speeds & Feeds Mill/Drill



The Milling/Drilling section of the Speed/Feed area will provide the user with the proper speed and feed data for milling and drilling various materials.

Standard Use:

1) Select the cutting tool material.
2) Enter the tool diameter in either inch or metric units in the appropriate field and press "ENTER". The diameter in the other unit of measure will be generated.
3) Select the workpiece material from the list.
4) Make any necessary changes to the "Load per Flute" and " No. of Flutes"
5) Click on the "Calculate" button and the proper results are generated.
6) You can then modify any of the values (Diameter, Depth of pass etc.) and click on "Calculate" again to generate the updated results.

Speeds & Feeds Mill/Drill

High Speed Machining Options:

Machinist ToolBox® also provides the user with the option to drive the calculations by a percentage of the maximum spindle RPM, the Maximum IPM Feedrate, or a combination of both.

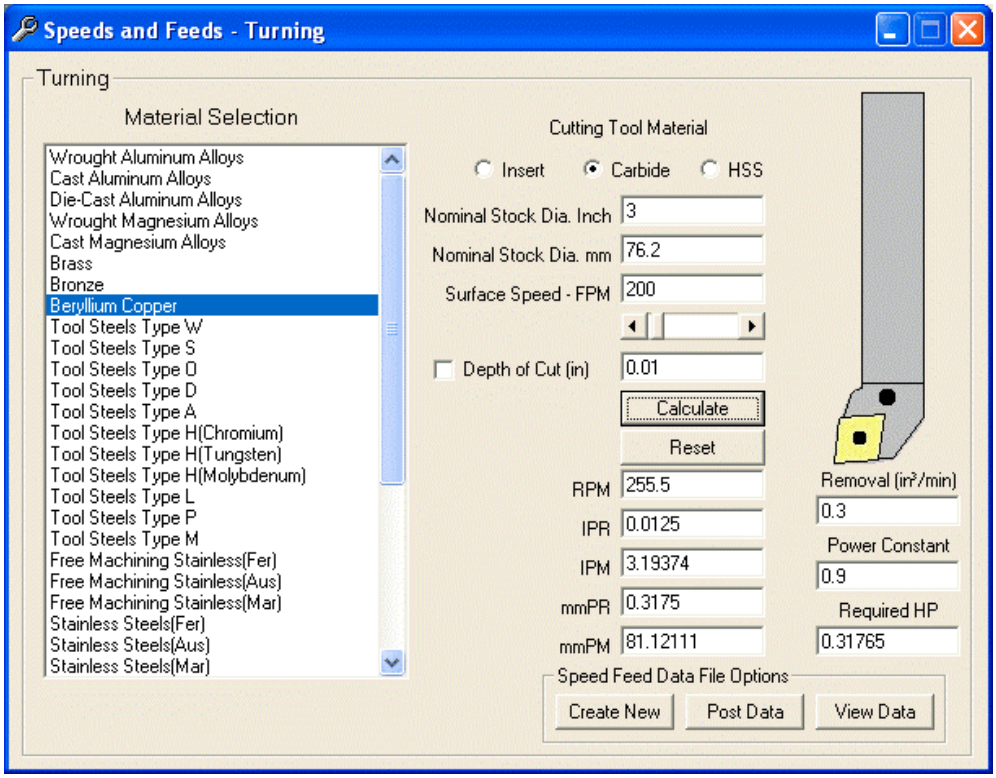
Since RPM and feedrate are functions resulting from the Surface Speed, it is necessary to show the resulting changes that were made by using this option. To that end, the color field legend is provided to show the user what the actual values for Surface Speed and Chip load are.

After each High Speed calculation, the user should click the "Reset" button and begin with fresh data.

The results may be saved to a data file that can be saved, printed and issued to the necessary individuals or saved for future reference.

IMPORTANT NOTE!
ONLY those who are familiar with the concept of High Speed Machining should use the figures generated by High Speed Machining option. Improper use of High Speed data may result in damage to the cutting tool, workpiece and machine tool. Any failure of these at High Speed may also endanger the safety of the machine tool operator.

Speeds & Feeds Turning

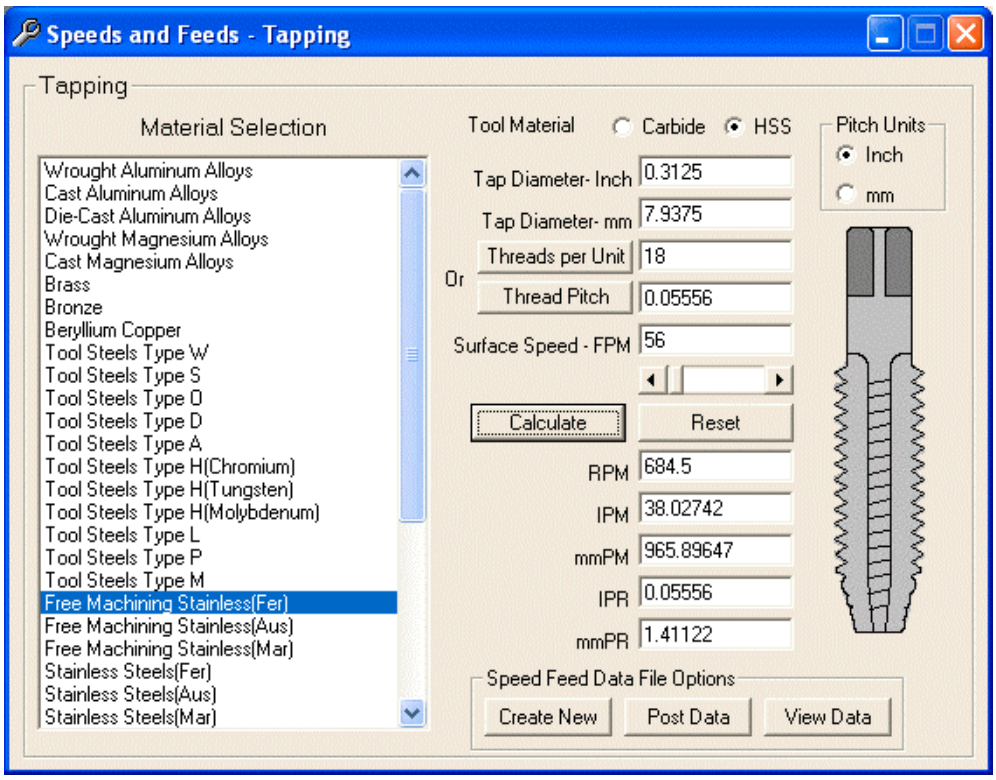


The Turning section of the Speed/Feed area provides the user with the proper speed and feed data for turning various materials.

1) Select the cutting tool material.
2) Enter the nominal stock diameter in either inch or metric units in the appropriate field and press "ENTER". The diameter in the other unit of measure will be generated.
3) Select the workpiece material from the list.
4) Click on the "Calculate" button and the proper results are generated.
5) You can then modify any of the values and click on "Calculate" again to generate the updated results.

The results may be saved to a data file that can be saved, printed and issued to the necessary individuals or saved for future reference.

Speeds & Feeds Tapping

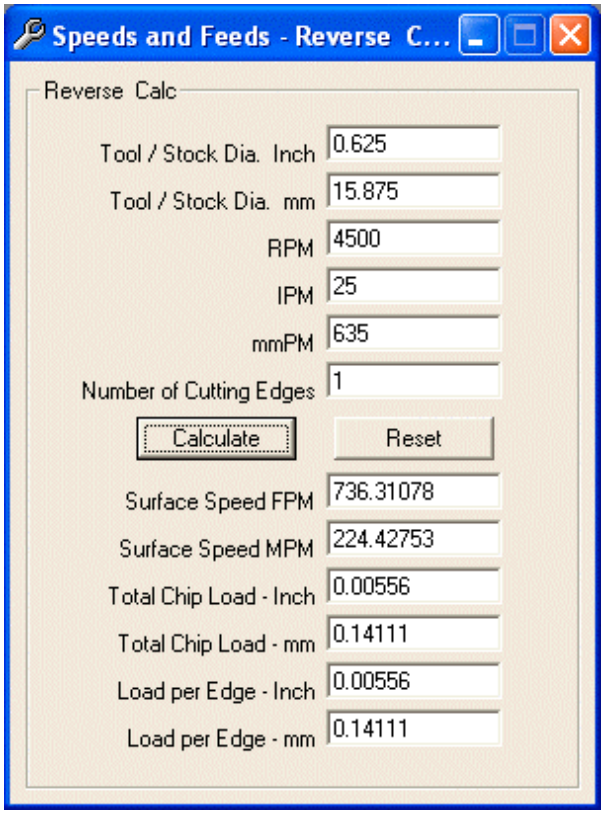


The Tapping section of the Speed/Feed area provides the user with the proper speed and feed data for tapping various materials.

1) Select the Tap material
2) Select the Pitch units (inch or mm)
3) Enter either the pitch or Threads per unit, click on the button next to that value and the inverse value will be calculated.
4) Enter the tap diameter in either inch or metric units in the appropriate field and press "ENTER". The tap diameter in the other unit of measure will be generated.
5) Select the workpiece material from the list.
6) Click on the "Calculate" button and the proper results are generated.

The results may be saved to a data file that can be saved, printed and issued to the necessary individuals or saved for future reference.

Speeds & Feeds Reverse Lookup

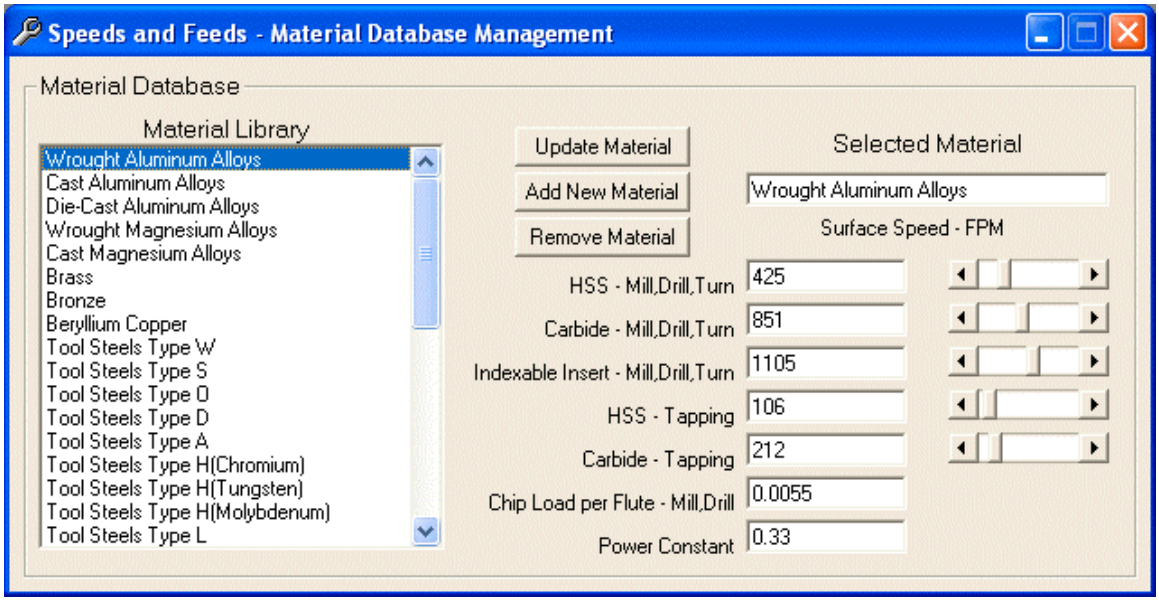


Many times, good old-fashioned trial and error is used for "dialing in" proper speeds and feeds.

With this in mind, the Reverse Look-Up function is very helpful when you need to know what your actual surface speed and chip load are for future reference.

With this data you can then modify the material database and eliminate
the trial and error method of getting proper speed and feed data.

Speeds & Feeds Material Database



Machinist ToolBox® allows the user to customize the default material database for calculating Speeds & Feeds and related data.

While the default material database was put together with great care, it was also put together on the conservative side. That is, the values generated may be slow for some users. The values for the default material database were calculated using a generic CNC knee mill as a guide. This option is provided, as there are many variables that can affect the actual speed and feed data used for a given application.

The user can adjust the various surface speeds by simply sliding or clicking on the scroll bars for each area. The user can also change the text that is shown in the material list as well as the default chip load and material power constant.

Updating a Material

1) Select a material to be updated by clicking on that material in the list.
2) Make all your necessary changes to surface speed, chip load, power constant and the material text for the list.
3) Click on the "Update Material" button and your changes to the database will be saved.

Adding a NEW Material

1) Select a material that is close to the new one you wish to create by clicking on that material in the list.
2) Make all your necessary changes to surface speed, chip load, and power constant. Type the NEW Material text for the list in the appropriate field.
3) Click on the "Add New Material" button and your new material will be added to the database.

Removing a Material

1) Select the material to be removed by clicking on that material in the list.
2) Click on the "Remove Material" button and the selected material will be removed from the database.

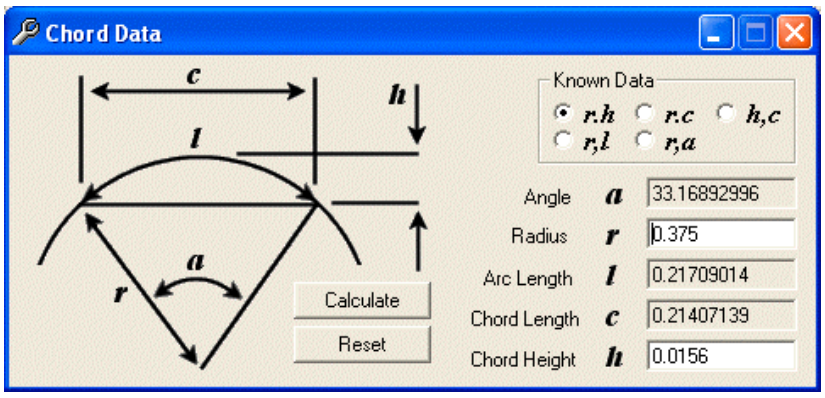
Trig Calculations

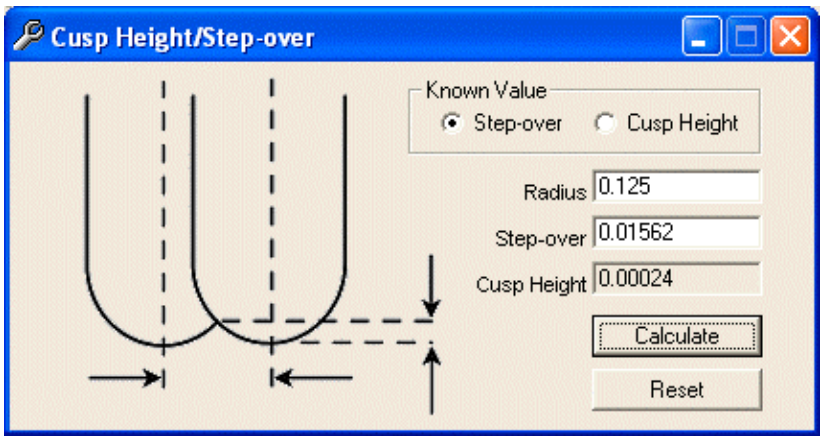
Machinist ToolBox® supports the following trigonometric calculations.

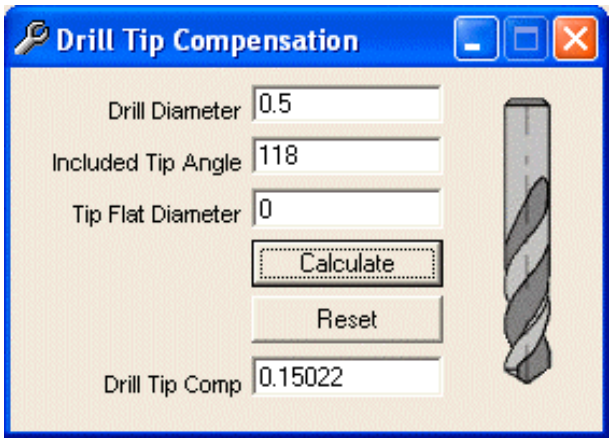
- **Chord Data**
- **Cusp Height/Step-over**
- **Drill Tip Compensation**
- **Sine Plate**
- **True Position**
- **Three Point Arc Center**
- **Right Triangle Solver**
- **Oblique Triangle Solver**
- **Bolt Circle Calculator**
- **Polygon Calculator**
- **Fourth Axis Positioning**
- **Universal Angle Head**
- **Thread Measurement**
- **Polar Dimensioning**

These are found by selecting "Trig Calculations" from the main menu.

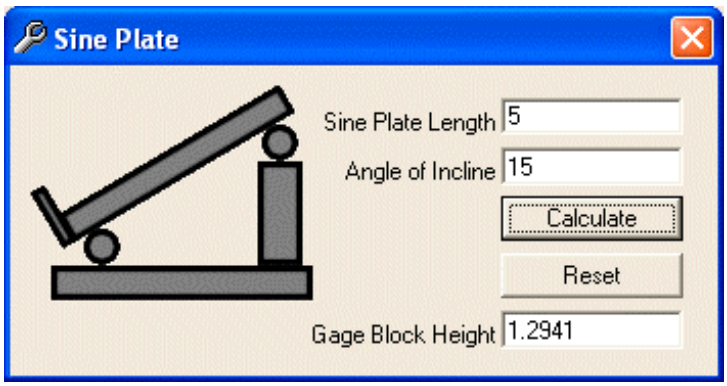
Trig Calculations

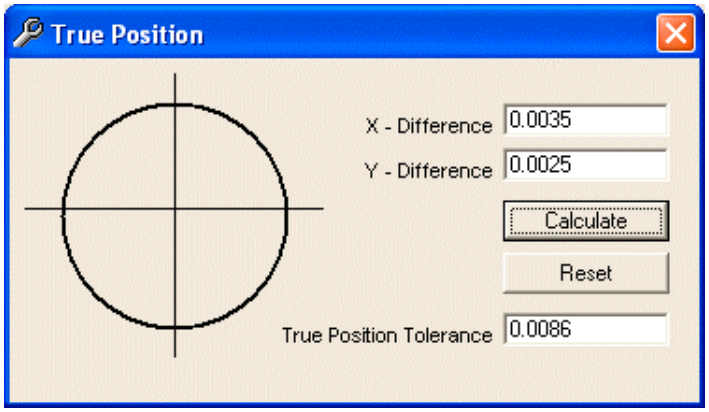


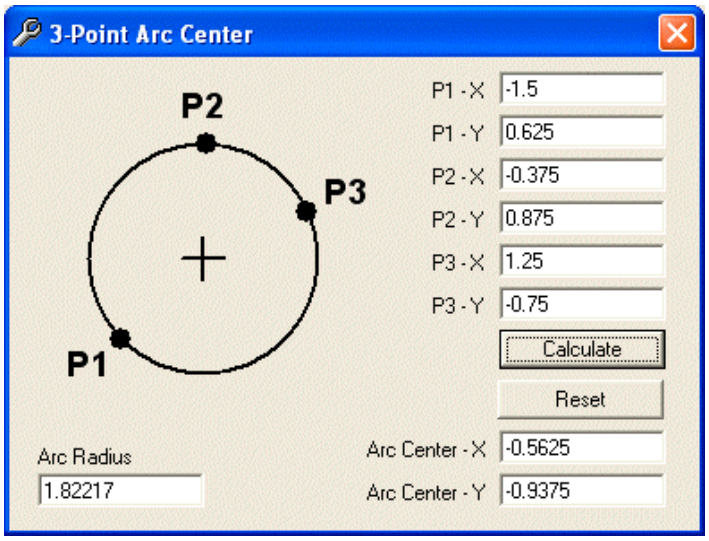




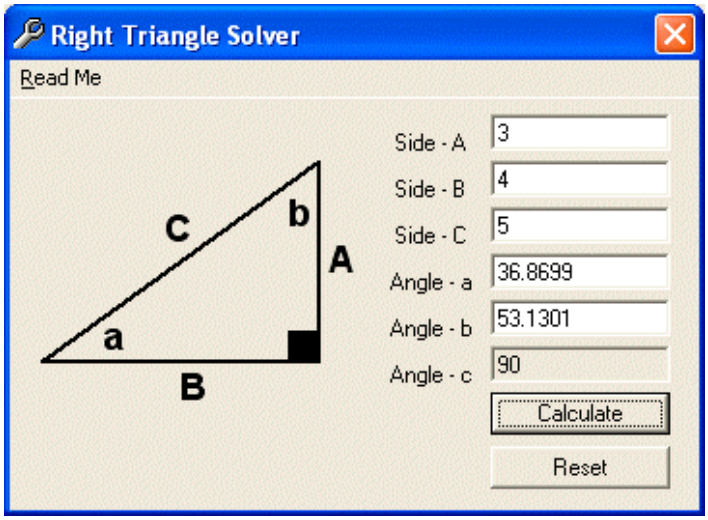
Trig Calculations

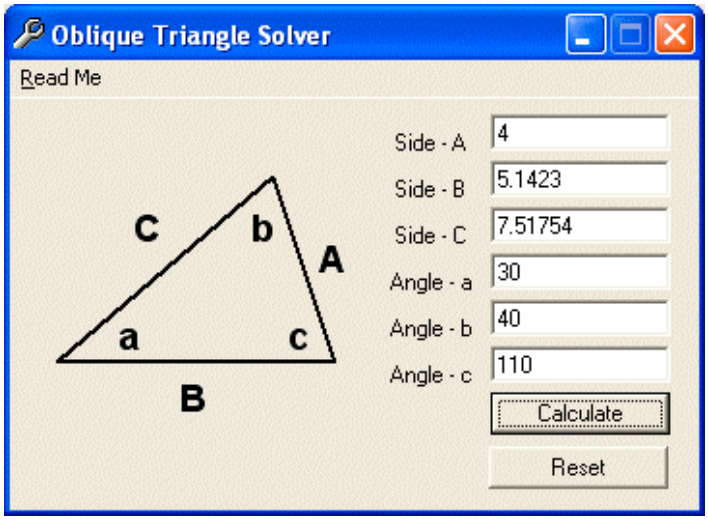


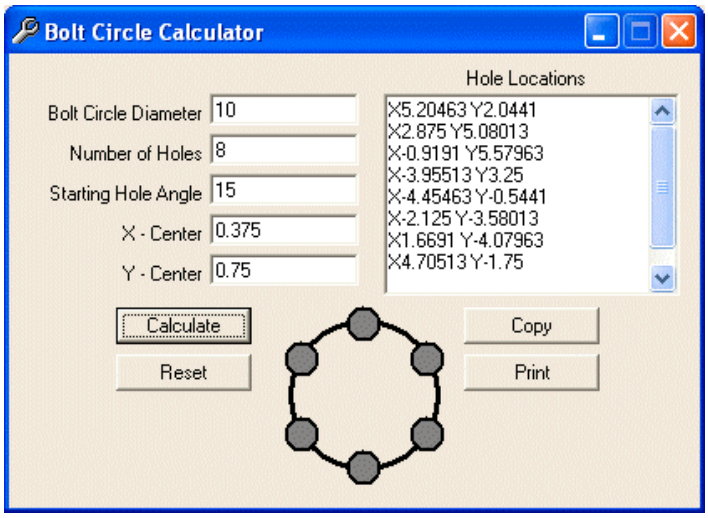




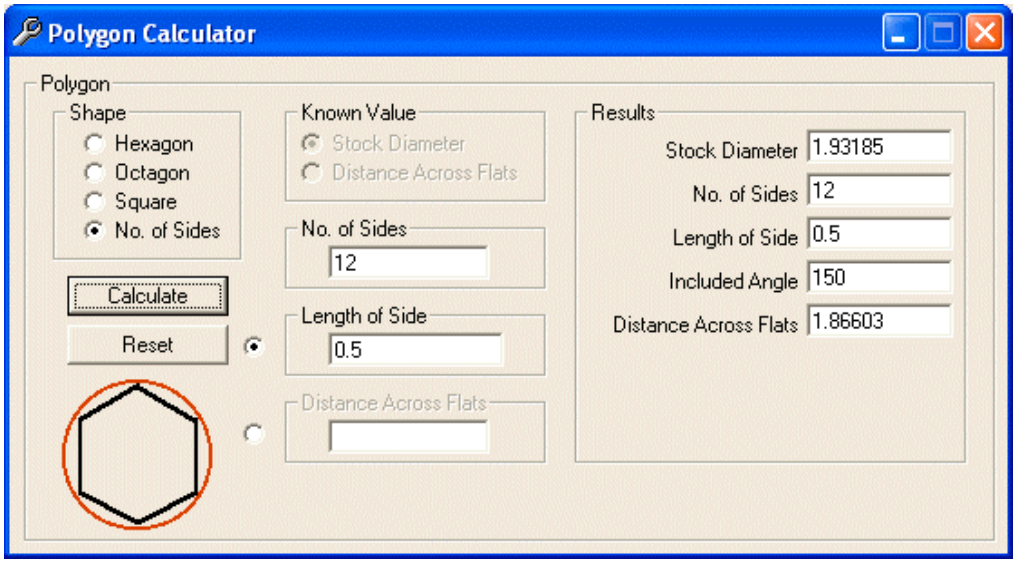
Trig Calculations

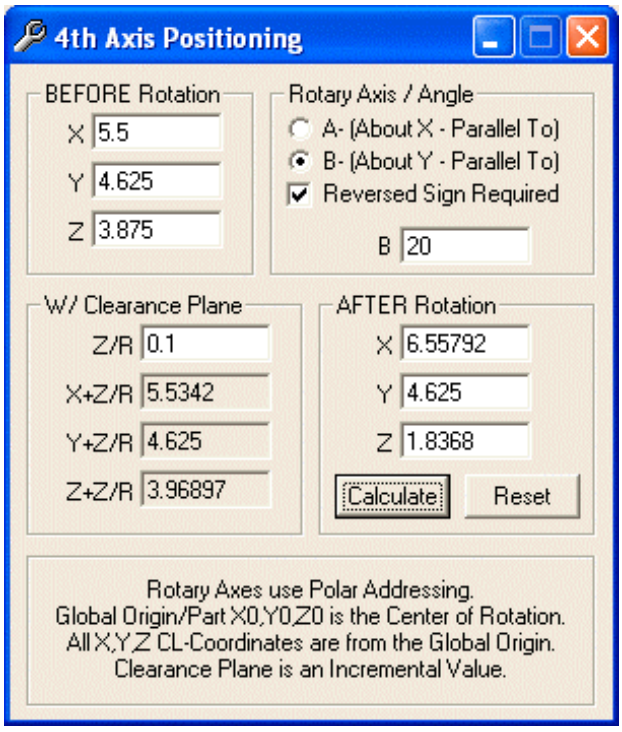




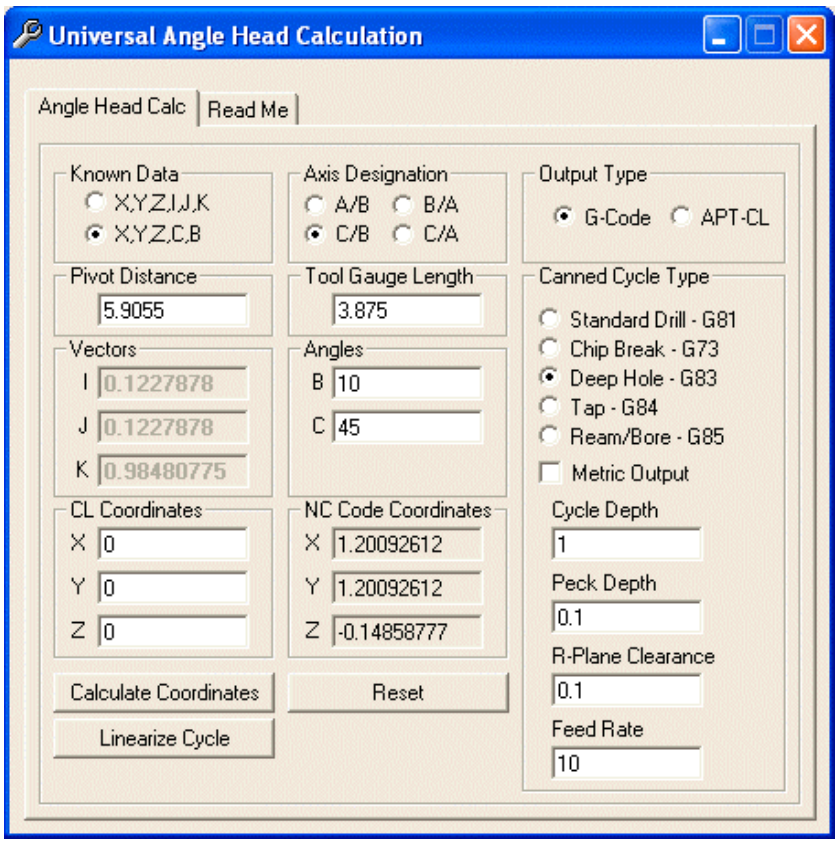


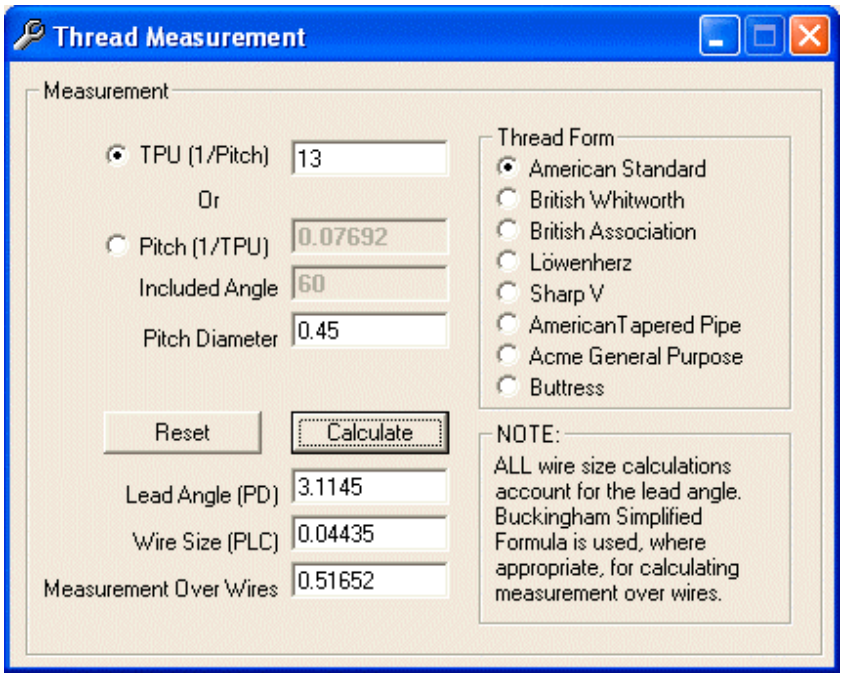
Trig Calculations



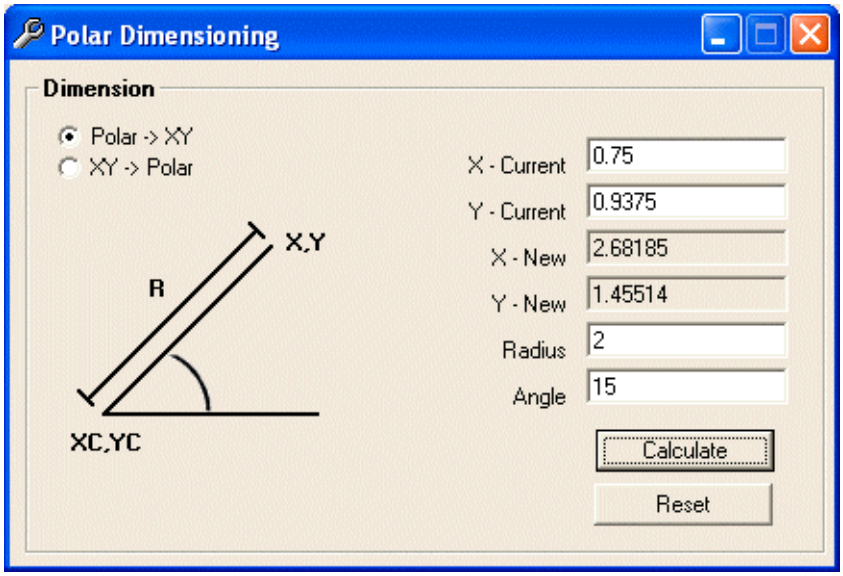


Trig Calculations

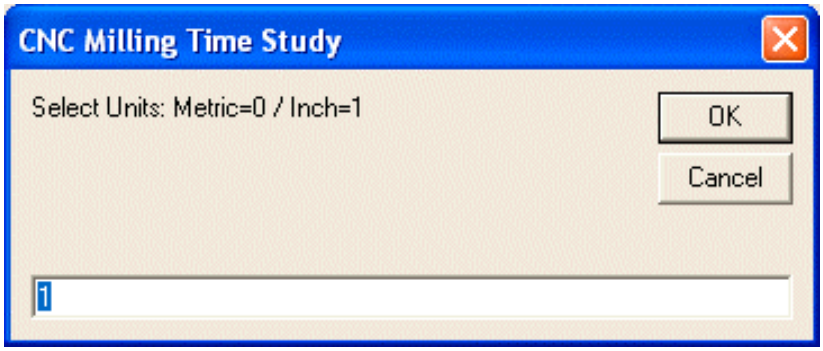


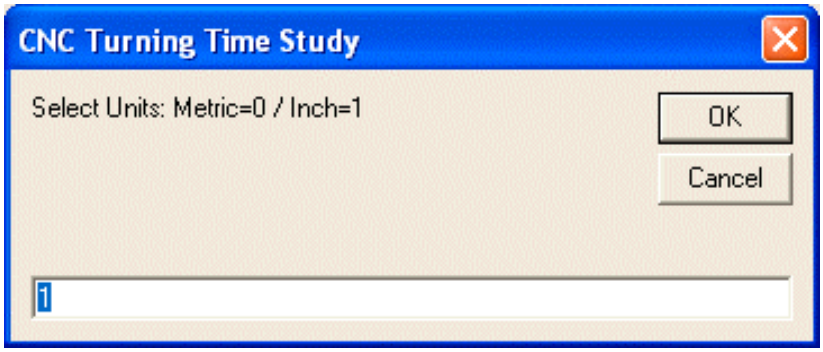


Trig Calculations



Time Studies Mill/Drill, Turn

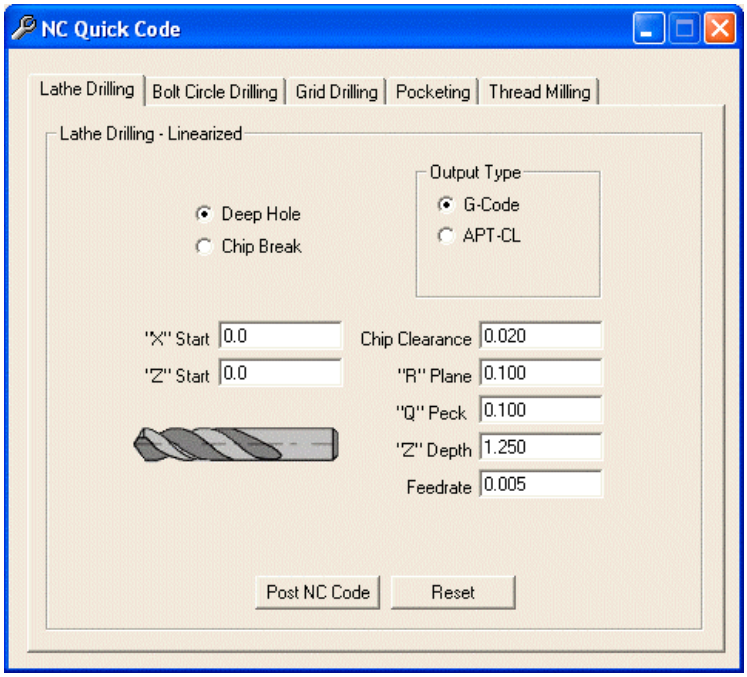


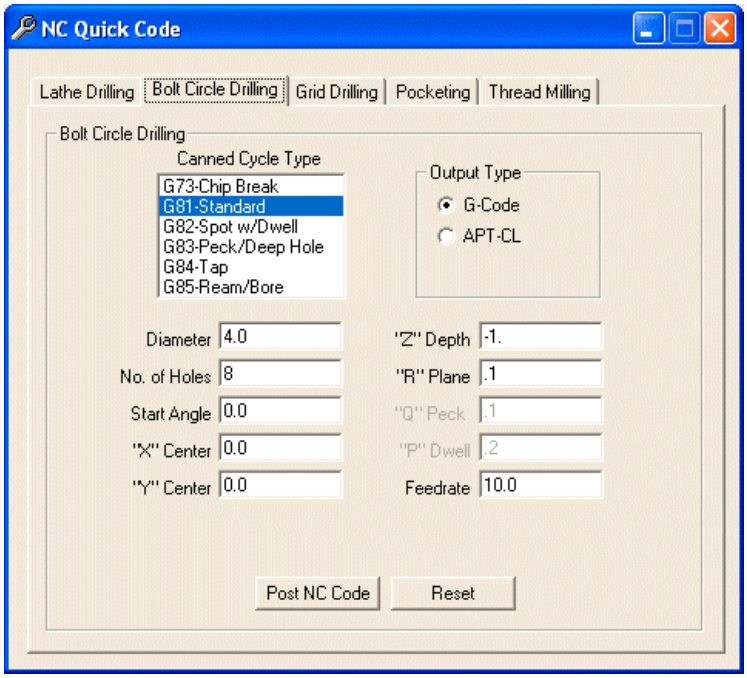


NC Quick Code Calculations

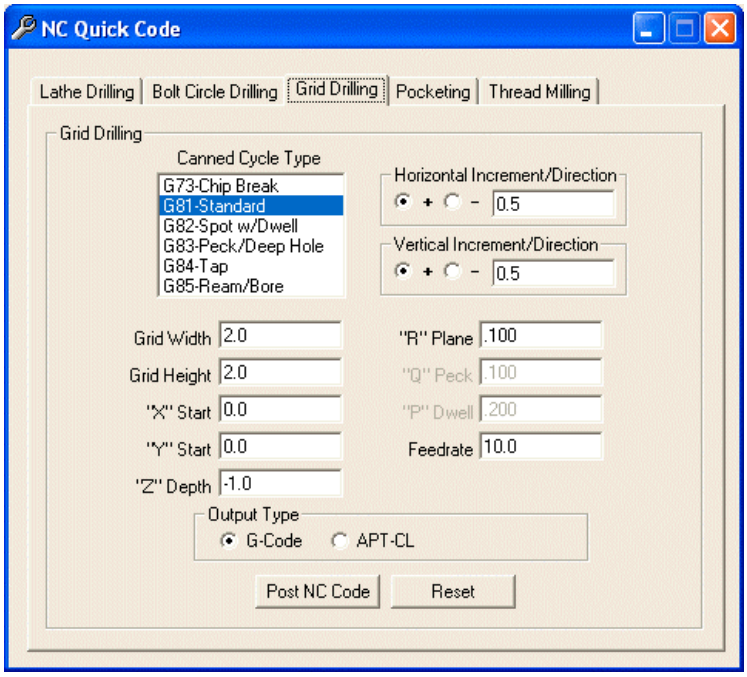
Machinist ToolBox® supports the following NC Quick Code calculations.

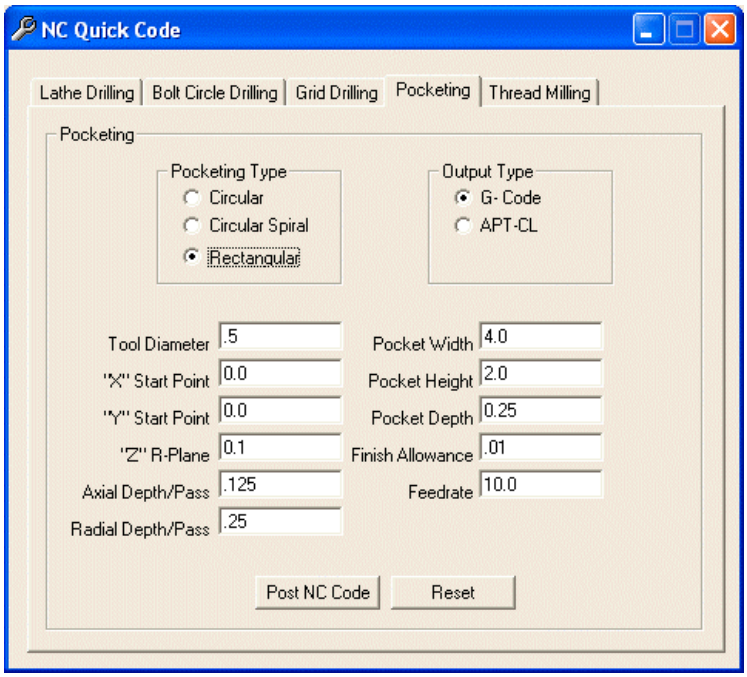
- **Lathe Drilling - Linearized**
- **Bolt Circle Drilling Cycles**
- **Grid Drilling Cycles**
- **Pocketing Operations**
- **Thread Milling Operations**



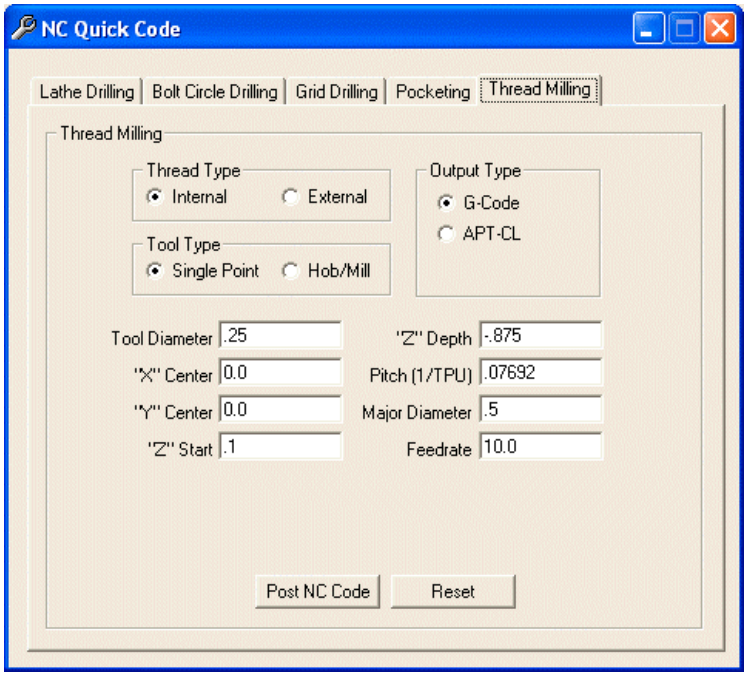


NC Quick Code Calculations

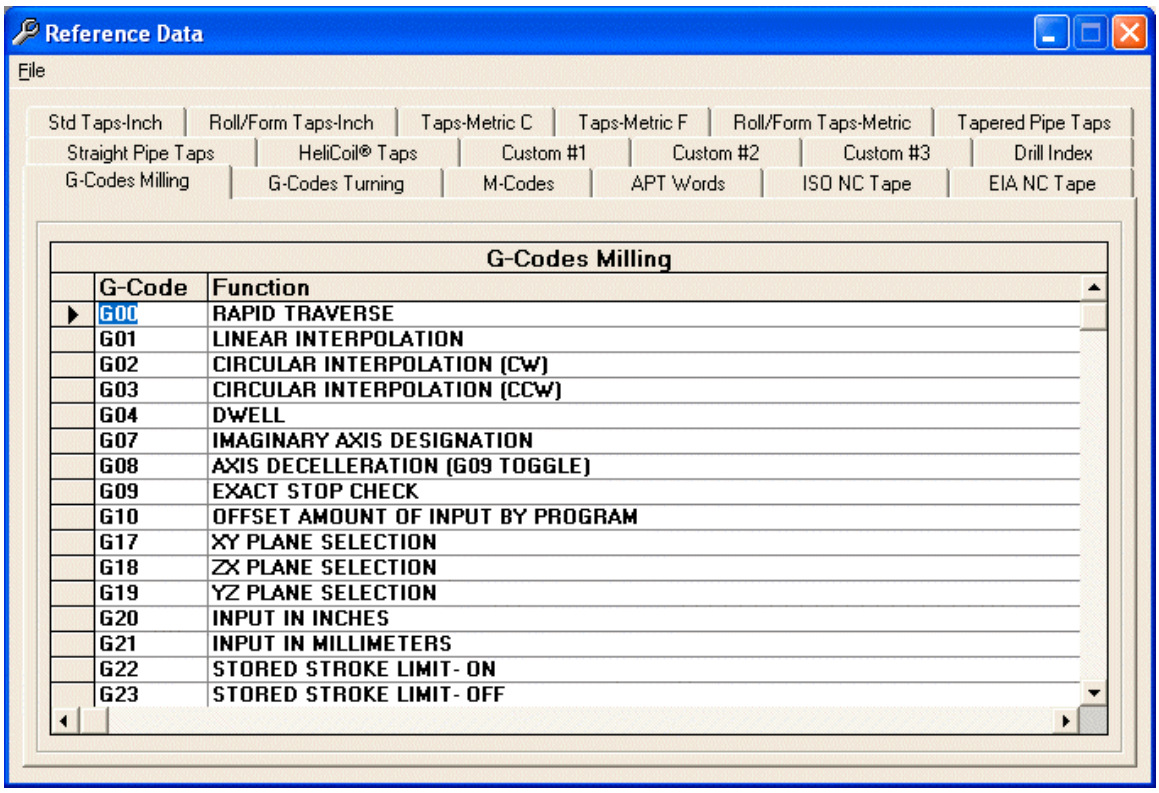




NC Quick Code Calculations



Reference Data



The Reference date area of Machinist ToolBox® has been updated to use a database grid format. It has also been updated and now includes the following:

- **Milling G-Codes**
- **Turning G-Codes**
- **M-Codes**
- **APT Standard words**
- **ISO NC Tape code**
- **EIA NC Tape code**
- **Straight Pipe Tap data**
- **Tapered Pipe Tap data**
- **HeliCoil® Tap data**
- **Standard cutting Tap data (inch)**
- **Roll/Form Tap data (inch)**
- **Standard Metric Tap data**
- **Metric Roll/Form Tap data**
- **3 Custom Chart Areas**

  It is now possible to customize your Milling & Turning G-Codes and M-Codes with machine specific data. You may also create 3 completely user specific charts for use on this dialogue.