

Machinist ToolBox® v7.1.3

Distributed by Bair Technologies

E-Mail: sales@ebair.com

WWW: <http://www.ebair.com>

Copyright© 1999-2004 CNC Machinist Software



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 screenshot shows a software window titled "Conversions" with a blue title bar and standard window controls. The window is divided into several sections, each with a title and a list of radio button options. Below the options are input and output fields. The sections are:

- Inch 2 Metric Units:** Options include Inch to cm, mm, micron, and vice-versa. Input: 1 mm, Output: 0.03937 in.
- Inch 2 Metric Speed:** Options include Inch/min to mm/min, Ft/min to M/min, mm/min to Inch/min, and M/min to Ft/min. Input: 1 ipm, Output: 25.4 mmpm.
- Dimension:** Options include Polar to XY and XY to Polar. Input: 1.5, 15, Output: 1.44889, 0.38823.
- Force/Torque:** Options include lbs to Kg, Tons to Metric Tons, lbs-ft to Newton Meters, Kg to lbs, Metric Tons to Tons, and Newton Meters to lbs-ft. Input: 10 lb, Output: 4.53592 kg.
- Angles:** Options include Degrees to Radians and Radians to Degrees. Input: 1 radians, Output: 57.29578 degrees.
- Volume:** Options include Cubic Inch to Liter and Liter to Cubic Inch. Input: 302 in³, Output: 4.94895 liters.
- Power:** Options include Hp to kW and kW to hp. Input: 15 hp, Output: 11.1855 kw.
- Pressure:** Options include psi to bars, psi to N/mm², bar to psi, and N/mm² to psi. Input: 15 lbs/in², Output: 1.03421 bars.
- NC Tape / Memory:** Options include ft to bytes, Meter to bytes, characters to bytes, bytes to characters, ft to characters, Meter to characters, characters to ft, bytes to ft, ft to Meter, Meter to ft, characters to Meter, and bytes to Meter. Input: 120 bytes, 1 ft.
- Temperature / Degrees:** Options include Fahrenheit to Celsius, Celsius to Fahrenheit, Celsius to Kelvin, and Fahrenheit to Rankine. Input: 32 °F, Output: 0 °C.

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 convert 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.

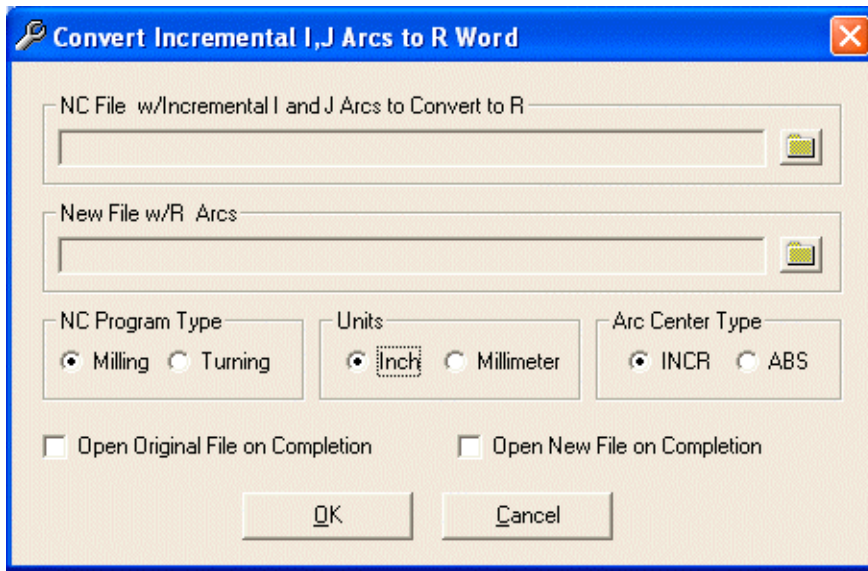
NC Tape/Memory

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

Mill / Drill

Material Selection

Wrought Aluminum Alloys
Cast Aluminum Alloys
Die-Cast Aluminum Alloys
Wrought Magnesium Alloys
Cast Magnesium Alloys
Brass
Bronze
Beryllium Copper
Tool Steels Type W
Tool Steels Type S
Tool Steels Type O
Tool Steels Type D
Tool Steels Type A
Tool Steels Type H(Chromium)
Tool Steels Type H(Tungsten)
Tool Steels Type H(Molybdenum)
Tool Steels Type L
Tool Steels Type P
Tool Steels Type M
Free Machining Stainless(Fer)
Free Machining Stainless(Aus)
Free Machining Stainless(Mar)
Stainless Steels(Fer)

Speed Feed Data File Options

Create New Post Data View Data

Field Legend

Standard Machining
High Speed Machining
Modified Result of HSM Calculation
Affects Removal (in³/min) + HP Update

Tool Material

Insert Carbide HSS

Tool Diameter-Inch 0.5
Tool Diameter-mm 12.7

Surface Speed - FPM 425

Load per Flute (in) 0.0055
No. of Flutes 3

High Speed Machining Options

Spindle RPM Override
 Drive by % of Max Enter Max RPM
N/A

Feedrate Override
 Force To IPM? 200

Calculate Reset

Power Constant 0.33
Removal (in³/min) 4.55358
Required HP 1.76786
Update

RPM 3246.8
IPM 53.57155
mmPM 1360.71747
Max. Radial DOC 0.2
Max. Axial DOC 0.425

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

Speeds and Feeds - Turning

Turning

Material Selection

- Wrought Aluminum Alloys
- Cast Aluminum Alloys
- Die-Cast Aluminum Alloys
- Wrought Magnesium Alloys
- Cast Magnesium Alloys
- Brass
- Bronze
- Beryllium Copper**
- Tool Steels Type W
- Tool Steels Type S
- Tool Steels Type O
- Tool Steels Type D
- Tool Steels Type A
- Tool Steels Type H(Chromium)
- Tool Steels Type H(Tungsten)
- Tool Steels Type H(Molybdenum)
- Tool Steels Type L
- Tool Steels Type P
- Tool Steels Type M
- Free Machining Stainless(Fer)
- Free Machining Stainless(Aus)
- Free Machining Stainless(Mar)
- Stainless Steels(Fer)
- Stainless Steels(Aus)
- Stainless Steels(Mar)

Cutting Tool Material

Insert Carbide HSS

Nominal Stock Dia. Inch 3

Nominal Stock Dia. mm 76.2

Surface Speed - FPM 200

Depth of Cut (in) 0.01

Calculate

Reset

RPM 255.5

IPR 0.0125

IPM 3.19374

mmPR 0.3175

mmPM 81.12111

Removal (in³/min) 0.3

Power Constant 0.9

Required HP 0.31765

Speed Feed Data File Options

Create New Post Data View Data

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

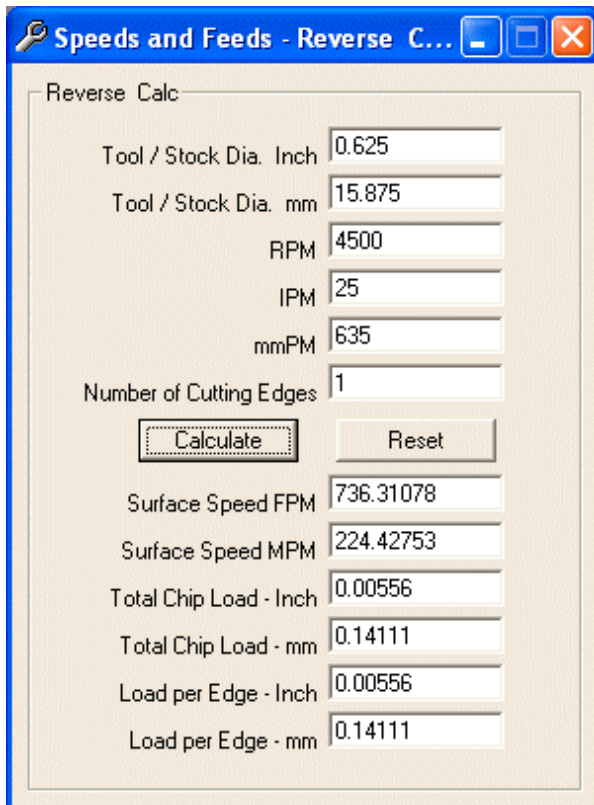
The screenshot shows the 'Speeds and Feeds - Tapping' software interface. On the left is a 'Material Selection' list with 'Free Machining Stainless(Fer)' selected. The 'Tool Material' is set to 'HSS'. The 'Pitch Units' are set to 'Inch'. The 'Tap Diameter- Inch' is 0.3125, and the 'Tap Diameter- mm' is 7.9375. The 'Threads per Unit' is 18, and the 'Thread Pitch' is 0.05556. The 'Surface Speed - FPM' is 56. The 'Calculate' button is highlighted. The resulting values are: RPM 684.5, IPM 38.02742, mmPM 965.89647, IPR 0.05556, and mmPR 1.41122. At the bottom, there are buttons for 'Create New', 'Post Data', and 'View Data'. A drawing of a tap is shown on the right side of the window.

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



The screenshot shows a software window titled "Speeds and Feeds - Reverse" with a "Reverse Calc" section. It contains several input fields for tool diameter, RPM, IPM, mmPM, and number of cutting edges, along with "Calculate" and "Reset" buttons. Below the buttons are output fields for surface speed (FPM and MPM) and total chip load (Inch and mm).

Input	Value
Tool / Stock Dia. Inch	0.625
Tool / Stock Dia. mm	15.875
RPM	4500
IPM	25
mmPM	635
Number of Cutting Edges	1

Buttons: Calculate, Reset

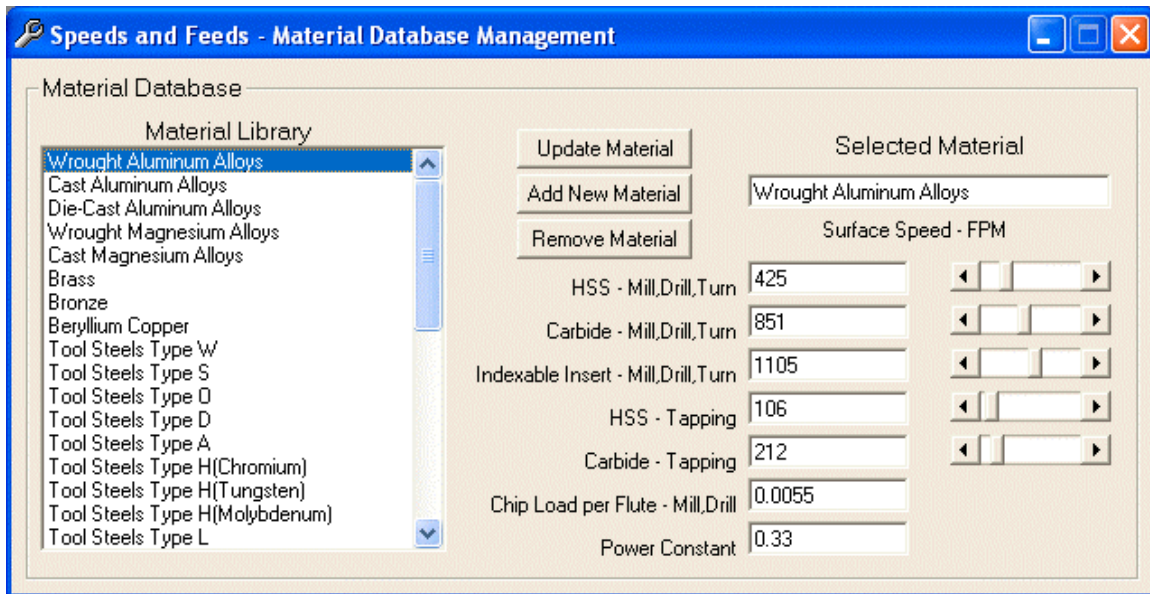
Output	Value
Surface Speed FPM	736.31078
Surface Speed MPM	224.42753
Total Chip Load - Inch	0.00556
Total Chip Load - mm	0.14111
Load per Edge - Inch	0.00556
Load per Edge - mm	0.14111

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

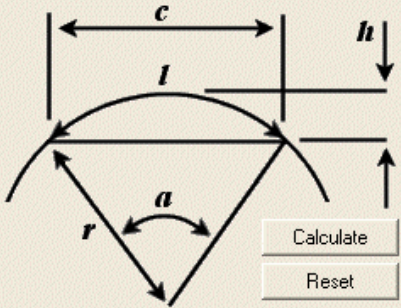
Machinist ToolBox® supports the following trigonometric calculations.

- **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

Chord Data



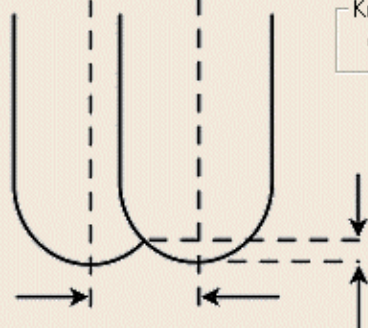
Known Data

r, h r, c h, c
 r, l r, a

Angle a 33.16892996
Radius r 0.375
Arc Length l 0.21709014
Chord Length c 0.21407139
Chord Height h 0.0156

Calculate
Reset

Cusp Height/Step-over



Known Value

Step-over Cusp Height

Radius 0.125
Step-over 0.01562
Cusp Height 0.00024


Calculate
Reset

Drill Tip Compensation

Drill Diameter 0.5
Included Tip Angle 118
Tip Flat Diameter 0

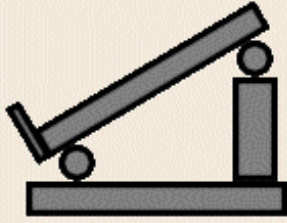
Calculate
Reset

Drill Tip Comp 0.15022



Trig Calculations

Sine Plate

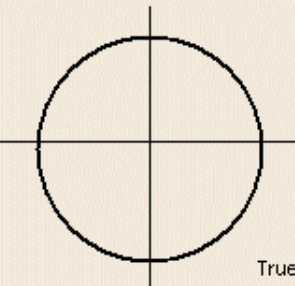


Sine Plate Length

Angle of Incline

Gage Block Height

True Position

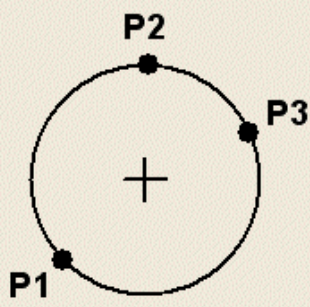


X - Difference

Y - Difference

True Position Tolerance

3-Point Arc Center



P1 - X

P1 - Y

P2 - X

P2 - Y

P3 - X

P3 - Y

Arc Radius

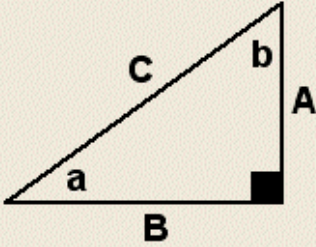
Arc Center - X

Arc Center - Y

Trig Calculations

Right Triangle Solver

Read Me

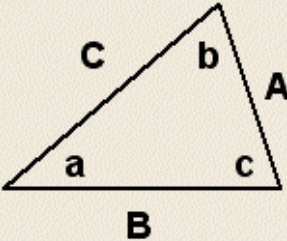


Side - A: 3
Side - B: 4
Side - C: 5
Angle - a: 36.8699
Angle - b: 53.1301
Angle - c: 90

Calculate
Reset

Oblique Triangle Solver

Read Me



Side - A: 4
Side - B: 5.1423
Side - C: 7.51754
Angle - a: 30
Angle - b: 40
Angle - c: 110

Calculate
Reset

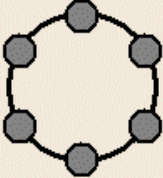
Bolt Circle Calculator

Hole Locations

Bolt Circle Diameter: 10
Number of Holes: 8
Starting Hole Angle: 15
X - Center: 0.375
Y - Center: 0.75

X5.20463 Y2.0441
X2.875 Y5.08013
X-0.9191 Y5.57963
X-3.95513 Y3.25
X-4.45463 Y-0.5441
X-2.125 Y-3.58013
X1.6691 Y-4.07963
X4.70513 Y-1.75

Calculate
Reset



Copy
Print

Trig Calculations

Polygon Calculator

Polygon

Shape

- Hexagon
- Octagon
- Square
- No. of Sides

Known Value

- Stock Diameter
- Distance Across Flats

No. of Sides:

Length of Side:

Distance Across Flats:

Results

Stock Diameter:

No. of Sides:


Length of Side:

Included Angle:

Distance Across Flats:

Calculate

Reset



4th Axis Positioning

BEFORE Rotation

X:

Y:

Z:

Rotary Axis / Angle

- A- (About X - Parallel To)
- B- (About Y - Parallel To)
- Reversed Sign Required

B:

W/ Clearance Plane

Z/R:

X+Z/R:

Y+Z/R:

Z+Z/R:

AFTER Rotation

X:

Y:

Z:

Calculate

Reset

Rotary Axes use Polar Addressing.
Global Origin/Part X0,Y0,Z0 is the Center of Rotation.
All X,Y,Z CL-Coordinates are from the Global Origin.
Clearance Plane is an Incremental Value.

Trig Calculations

Universal Angle Head Calculation

Angle Head Calc | Read Me

Known Data <input type="radio"/> X,Y,Z,I,J,K <input checked="" type="radio"/> X,Y,Z,C,B	Axis Designation <input type="radio"/> A/B <input type="radio"/> B/A <input checked="" type="radio"/> C/B <input type="radio"/> C/A	Output Type <input checked="" type="radio"/> G-Code <input type="radio"/> APT-CL
Pivot Distance 5.9055	Tool Gauge Length 3.875	Canned Cycle Type <input type="radio"/> Standard Drill - G81 <input type="radio"/> Chip Break - G73 <input checked="" type="radio"/> Deep Hole - G83 <input type="radio"/> Tap - G84 <input type="radio"/> Ream/Bore - G85 <input type="checkbox"/> Metric Output
Vectors I 0.1227878 J 0.1227878 K 0.98480775	Angles B 10 C 45	Cycle Depth 1 Peck Depth 0.1 R-Plane Clearance 0.1 Feed Rate 10
CL Coordinates X 0 Y 0 Z 0	NC Code Coordinates X 1.20092612 Y 1.20092612 Z -0.14858777	

Calculate Coordinates Reset

Linearize Cycle

Thread Measurement

Measurement

<input checked="" type="radio"/> TPU (1/Pitch) 13 Or <input type="radio"/> Pitch (1/TPU) 0.07692 Included Angle 60 Pitch Diameter 0.45	Thread Form <input checked="" type="radio"/> American Standard <input type="radio"/> British Whitworth <input type="radio"/> British Association <input type="radio"/> Löwenherz <input type="radio"/> Sharp V <input type="radio"/> American Tapered Pipe <input type="radio"/> Acme General Purpose <input type="radio"/> Buttress
--	---

Reset Calculate

Lead Angle (PD) 3.1145
Wire Size (PLC) 0.04435
Measurement Over Wires 0.51652

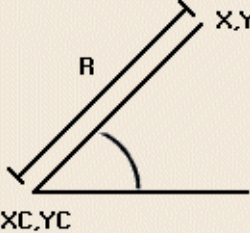
NOTE:
ALL wire size calculations account for the lead angle. Buckingham Simplified Formula is used, where appropriate, for calculating measurement over wires.

Trig Calculations

Polar Dimensioning

Dimension


Polar \rightarrow XY
 XY \rightarrow Polar




X - Current: 0.75
Y - Current: 0.9375
X - New: 2.68185
Y - New: 1.45514
Radius: 2
Angle: 15

Calculate
Reset

Time Studies Mill/Drill, Turn

CNC Milling Time Study 

Select Units: Metric=0 / Inch=1

CNC Turning Time Study 

Select Units: Metric=0 / Inch=1

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**

The screenshot shows the 'NC Quick Code' dialog box with the 'Lathe Drilling' tab selected. The 'Lathe Drilling - Linearized' section is active. It features two radio buttons for 'Deep Hole' (selected) and 'Chip Break'. The 'Output Type' section has 'G-Code' selected and 'APT-CL' unselected. Input fields include: 'X' Start (0.0), 'Z' Start (0.0), Chip Clearance (0.020), 'R' Plane (0.100), 'Q' Peck (0.100), 'Z' Depth (1.250), and Feedrate (0.005). A small image of a drill bit is shown. At the bottom are 'Post NC Code' and 'Reset' buttons.

The screenshot shows the 'NC Quick Code' dialog box with the 'Bolt Circle Drilling' tab selected. The 'Bolt Circle Drilling' section is active. It features a 'Canned Cycle Type' list with 'G81-Standard' selected. The 'Output Type' section has 'G-Code' selected and 'APT-CL' unselected. Input fields include: Diameter (4.0), 'Z' Depth (-1.0), No. of Holes (8), 'R' Plane (.1), Start Angle (0.0), 'Q' Peck (.1), 'X' Center (0.0), 'P' Dwell (.2), 'Y' Center (0.0), and Feedrate (10.0). At the bottom are 'Post NC Code' and 'Reset' buttons.

NC Quick Code Calculations

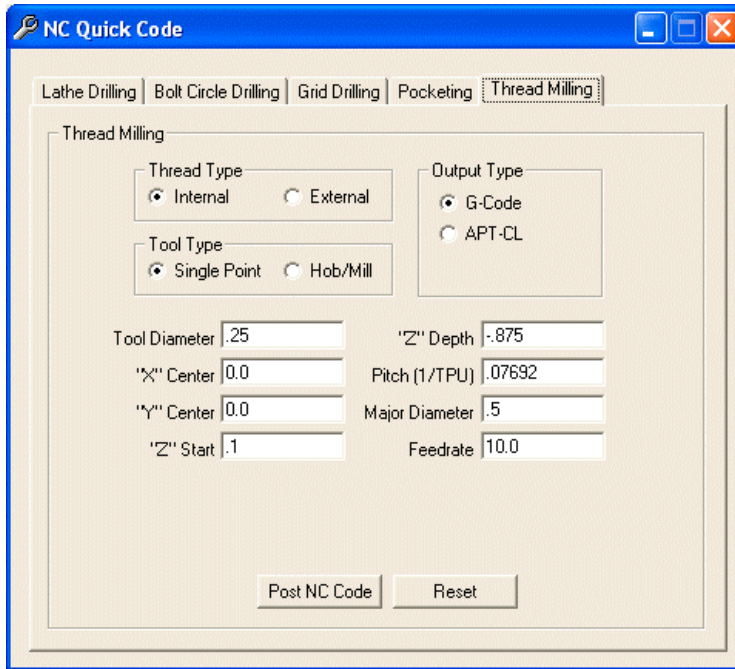
The screenshot shows the 'NC Quick Code' dialog box with the 'Grid Drilling' tab selected. The 'Canned Cycle Type' list includes G73-Chip Break, G81-Standard (selected), G82-Spot w/Dwell, G83-Peck/Deep Hole, G84-Tap, and G85-Ream/Bore. The 'Horizontal Increment/Direction' and 'Vertical Increment/Direction' are both set to 0.5. The 'Grid Width' and 'Grid Height' are both 2.0. The 'X' Start, 'Y' Start, and 'Z' Depth are 0.0, 0.0, and -1.0 respectively. The 'R' Plane is .100, 'Q' Peck is .100, and 'P' Dwell is .200. The 'Feedrate' is 10.0. The 'Output Type' is set to G-Code. Buttons for 'Post NC Code' and 'Reset' are at the bottom.

Parameter	Value
Canned Cycle Type	G81-Standard
Horizontal Increment/Direction	0.5
Vertical Increment/Direction	0.5
Grid Width	2.0
Grid Height	2.0
'X' Start	0.0
'Y' Start	0.0
'Z' Depth	-1.0
'R' Plane	.100
'Q' Peck	.100
'P' Dwell	.200
Feedrate	10.0
Output Type	G-Code

The screenshot shows the 'NC Quick Code' dialog box with the 'Pocketing' tab selected. The 'Pocketing Type' is set to Rectangular. The 'Output Type' is set to G-Code. The 'Tool Diameter' is .5, 'Pocket Width' is 4.0, 'X' Start Point is 0.0, 'Pocket Height' is 2.0, 'Y' Start Point is 0.0, 'Pocket Depth' is 0.25, 'Z' R-Plane is 0.1, 'Finish Allowance' is .01, 'Axial Depth/Pass' is .125, and 'Feedrate' is 10.0. The 'Radial Depth/Pass' is .25. Buttons for 'Post NC Code' and 'Reset' are at the bottom.

Parameter	Value
Pocketing Type	Rectangular
Output Type	G-Code
Tool Diameter	.5
Pocket Width	4.0
'X' Start Point	0.0
Pocket Height	2.0
'Y' Start Point	0.0
Pocket Depth	0.25
'Z' R-Plane	0.1
Finish Allowance	.01
Axial Depth/Pass	.125
Feedrate	10.0
Radial Depth/Pass	.25

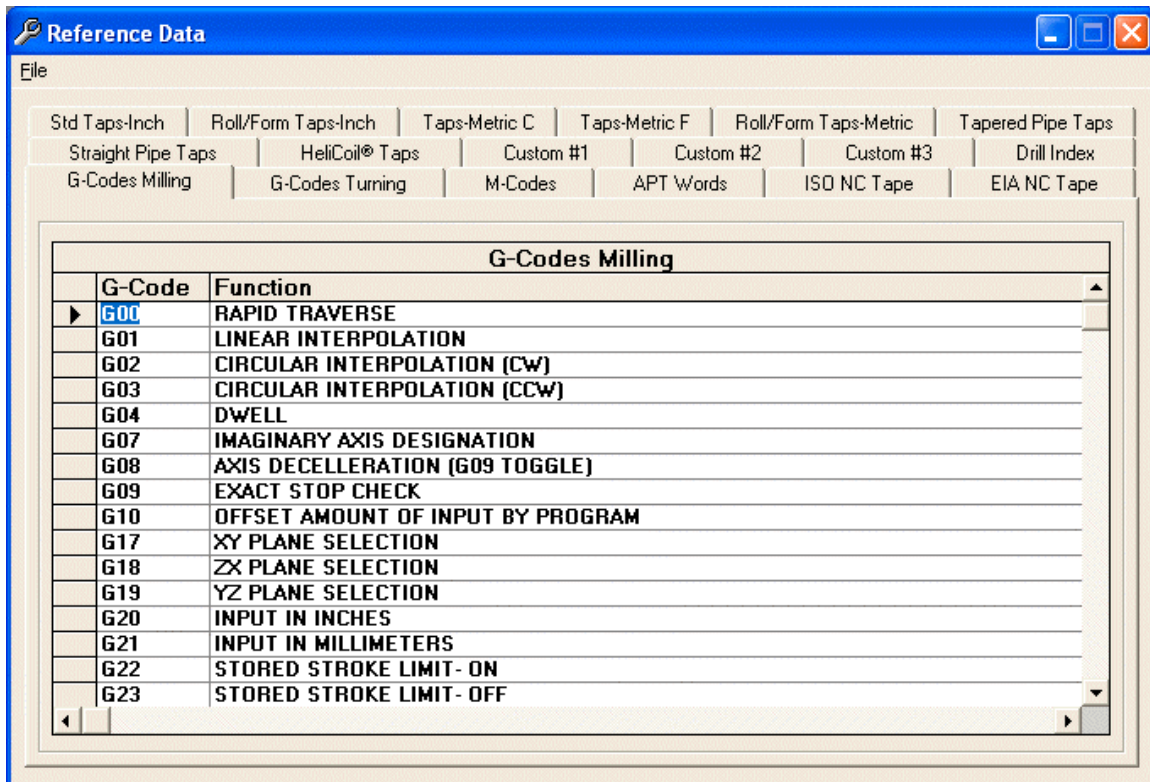
NC Quick Code Calculations



The screenshot shows the 'NC Quick Code' application window with the 'Thread Milling' tab selected. The interface includes several radio button options for 'Thread Type' (Internal, External), 'Tool Type' (Single Point, Hob/Mill), and 'Output Type' (G-Code, APT-CL). Below these are input fields for 'Tool Diameter', 'Z' Depth, 'X' Center, Pitch (1/TPU), 'Y' Center, Major Diameter, 'Z' Start, and Feedrate. At the bottom, there are 'Post NC Code' and 'Reset' buttons.

Parameter	Value
Tool Diameter	.25
'Z' Depth	-.875
'X' Center	0.0
Pitch (1/TPU)	.07692
'Y' Center	0.0
Major Diameter	.5
'Z' Start	.1
Feedrate	10.0

Reference Data



The Reference data 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.