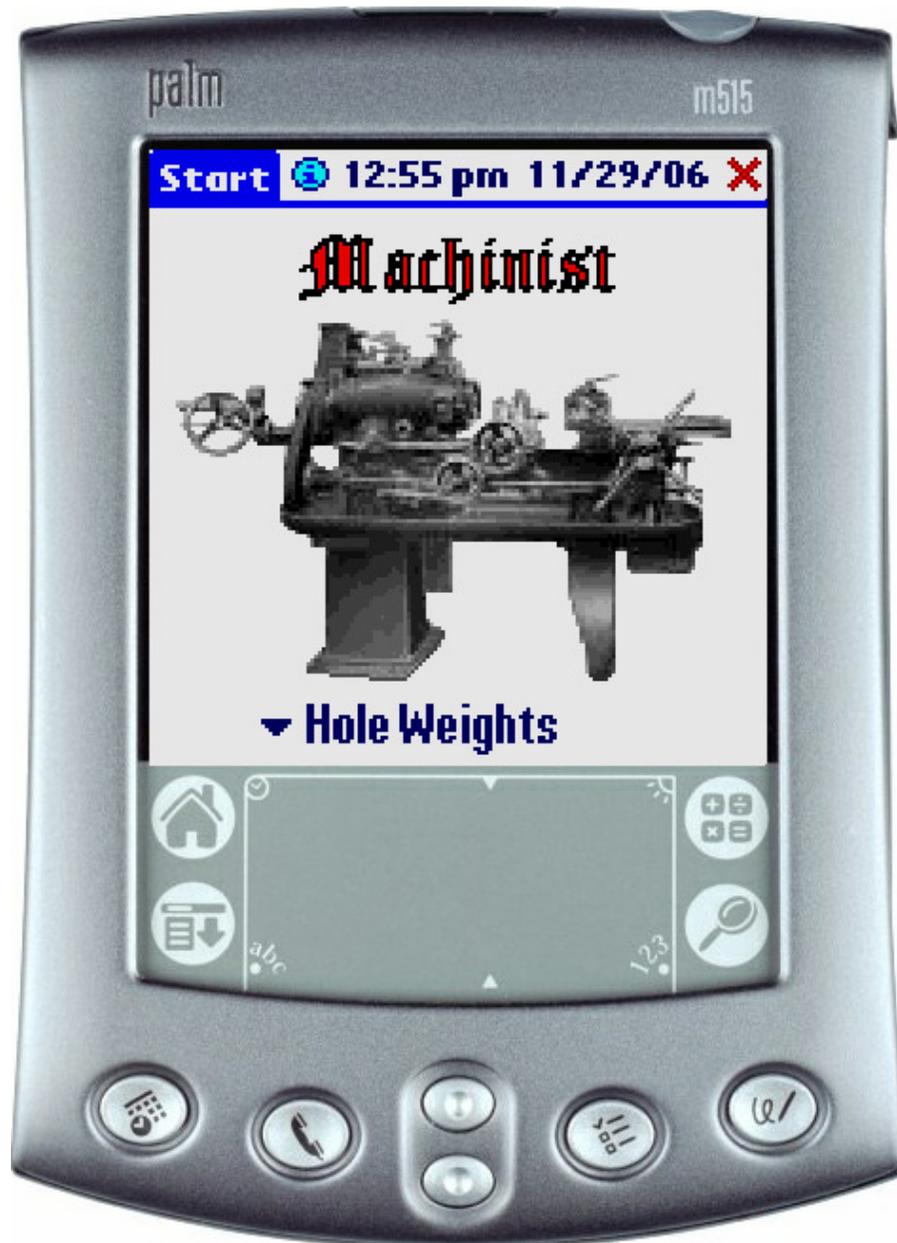


Machinist

User Guide



Machinist is **THE** premier Palm OS application for engineers, CNC programmers, CNC set-up, toolmakers, or anyone in the machining industry. **Machinist** is designed to ease CNC lathe programming of complex radius to angle to radius profiles. **Machinist** generates up to 60 bolt hole locations, regardless of start angle or position. **Machinist** simplifies drill tip depth compensation. **Machinist** generates CNC thread cycles, including turning, tapping, and milling. **Machinist** also calculates process cycle times and efficiency, plus much more. The highly graphical interface simplifies usage and aids in self explanation.

Minimum OS requirement: Any device with Palm OS 3.5 or higher (newer).

Machinist will operate on Palm OS's older than 3.5, but some functions may not work correctly.

Read the EULA section at the end of this User Guide before proceeding.

Hot sync **Machinist.prc** , **Machinist Timer.prc**, **Machinist Data.pdb**, & **MathLib.prc** onto your Palm powered device.

Note: Refer to your devices user manual for Hotsync instructions.

Machinist automatically installs into the MAIN directory.

The Demo version of **Machinist** is fully functional for 30 days. After the Demo has expired, a registered version of **Machinist** must be purchased.

Machinist has been compatibility tested on many Palm OS device; including Palm M100, M125, M130, M500, M505, III, VII, T3, Handspring Visor series, TRG, Sony Clie, Kyocera & Treo smart phones.

As of version 1.91, **Machinist** now supports all portrait and landscape mode Palm OS devices.

Tap the **Machinist** icon  to start the application.

Note: Some older Palm OS's do not display the time and date, on the main page. To get started, tap 'Start' or the down arrow under the lathe picture and select a category.

Note: Tap  at any time to access the extensive on board step by step User Guide.

Tip: Tap & hold any icon to view a short description of that buttons function.

Tap **X**, at any time, to exit **Machinist**.

Note: **Machinist** will automatically save all information that you enter, even after exiting the application.



Individual Application Screens

Bar Calc:

This App calculates the number of pieces that can be made from a bar of material.

Enter the **Bar Length** (in feet).

Enter the **Kerf/Cut-Off Width** (in inches).

Enter the **Remnant Length** (in inches).

Enter the bar diameter.

Enter the cut part length.

Select the material being used.

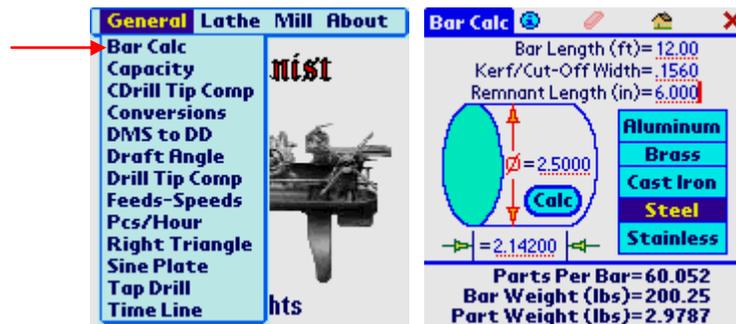
Tap **Calc** to figure the cut part and the bar weight.

Tap  at any time to access the onboard step by step instructions.

Tap  to clear your information.

Tap  to return to **Machinist** main.

Tap  to exit **Machinist**.



Capacity:

This App calculates the number of machines and cycle time required to run a product.

Enter the total customer **Annual Requirement** in pieces.

Enter the number of **Work Weeks In A Year**.

Enter the **Quoted Pcs Per Hour**.

Enter the number of **Days Per Week Worked**.

Enter the total number of **Hour Per Day Worked**.

Enter the **Number Of Machines** being utilized.

Tap **Calc** to calculate the requirements and cycle time needed to run the job.

Tap  at any time to access the onboard step by step instructions.

Tap  to clear your information.

Tap  to return to **Machinist** main.

Tap  to exit **Machinist**.



CDrill Tip Comp:

This App calculates the depth needed to create a specific chamfer size. Enter the required chamfer diameter.

- Tap **Calc** to calculate the depth needed to make the required chamfer diameter.
- Tap at any time to access the onboard step by step instructions.
- Tap to clear your information.
- Tap to return to **Machinist** main.
- Tap to exit **Machinist**.



Conversions:

This App converts a number from one category, into another.

TIP: Conversion is helpful on machinery that has metric pressure gages, as found on most CNC equipment.

Enter the **Value** to be converted.

- Tap **Calc** to calculate the conversion.
- Tap at any time to access the onboard step by step instructions.
- Tap to clear your information.
- Tap to return to **Machinist** main.
- Tap to exit **Machinist**.



DMS>DD:

This App converts angular dimensions between deg'min"sec" and decimal degrees.

TIP: DMS>DD is handy for converting conversational CNC program angles.

Top of the screen....

Enter the **degrees** of angle.

Enter the **minutes** of angle.

Enter the **seconds** of angle.

Middle of the page....

Enter the angle in **decimal degrees**.

Tap **Calc** to convert from angular degrees to decimal degrees, or vice versa.

Tap at any time to access the onboard step by step instructions.

Tap to clear your information.

Tap to return to **Machinist** main.

Tap to exit **Machinist**.



Draft Angle:

This App calculates the amount of draft angle in a casting, forging, etc.

Enter the draft Angle, in decimal degrees.

Enter the length of draft angle.

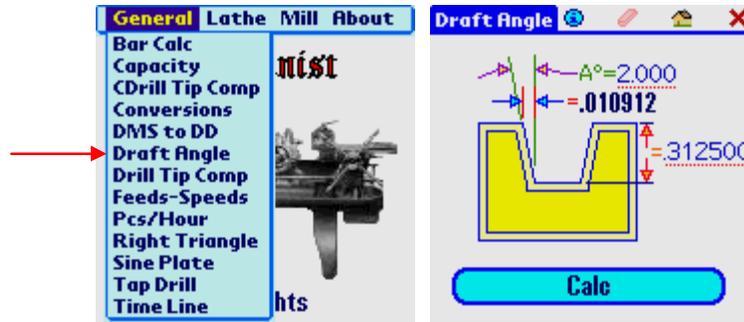
Tap **Calc** to calculate the amount of draft angle.

Tap at any time to access the onboard step by step instructions.

Tap to clear your information.

Tap to return to **Machinist** main.

Tap to exit **Machinist**.



Drill Tip Comp:

This App calculates the depth needed to compensate for a drill tip. Enter the drill diameter.

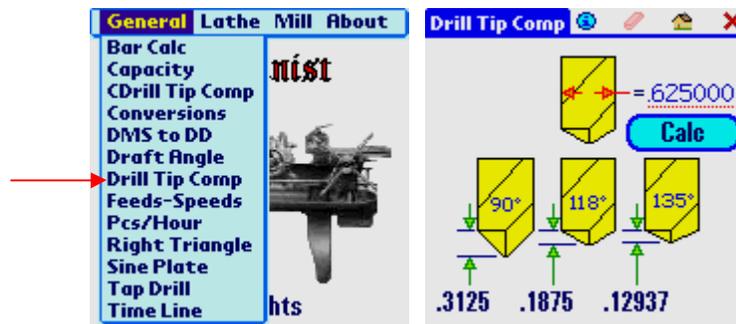
Tap **Calc** to calculate the drill point length.

Tap at any time to access the onboard step by step instructions.

Tap to clear your information.

Tap to return to **Machinist** main.

Tap to exit **Machinist**.



Feeds-Speeds:

This App calculates machining feeds and speeds.

TIP: Feeds-Speeds is a handy converter for all machinists and programmers.

To calculate **SFM** (surface feet per minute): Enter the **RPM** (revolutions per minute). Enter the tool **DIA**meter.

Tap **Calc** to calculate the **SFM**.

To calculate **RPM** (revolutions per minute): Enter the **SFM** (surface feet per minute). Enter the tool **DIA**meter.

Tap **Calc** to calculate the **RPM**.

To calculate **IPR** (inches per revolution): Enter the **IPM** (inches per minute) Enter the **RPM** (revolutions per minute).

Tap **Calc** to calculate the **IPR**.

To calculate **IPT** (inches per tooth): Enter the **IPM** (inches per minute). Enter the # **Teeth** (number of cutting edges). Enter the **RPM** (revolutions per minute).

Tap **Calc** to calculate **IPT**.

To calculate **IPM** (inches per minute): Enter the **IPT** (inches per tooth).

Enter the # **Teeth** (number of cutting edges).
 Enter the **RPM** (revolutions per minute).

Tap **Calc** to calculate **IPM**.

Tap  at any time to access the onboard step by step instructions.

Tap  to clear your information.

Tap  to return to **Machinist** main.

Tap  to exit **Machinist**.



Pcs/Hour:

This App calculates the number of parts per hour that a machine(s) can produce.

Tap  to use the stopwatch.

Tip: Tap the graffiti home icon to return to **Machinist** when finished with the stopwatch.

Enter the total machine **Cycle Time In Seconds** (start to start including load time).

Enter the **Number Of Parts Per Load**.

Enter the **Number Of Machines** being utilized.

Tip: Tap **Calc** now to only figure 100%, 90%, & 80% efficiencies.

Enter the total number of **Hours Per Day Worked**.

Enter the number of **Days Per Week Worked**.

Enter the number of **Work Weeks In A Year**.

Tap **Calc** to calculate the machine efficiencies.

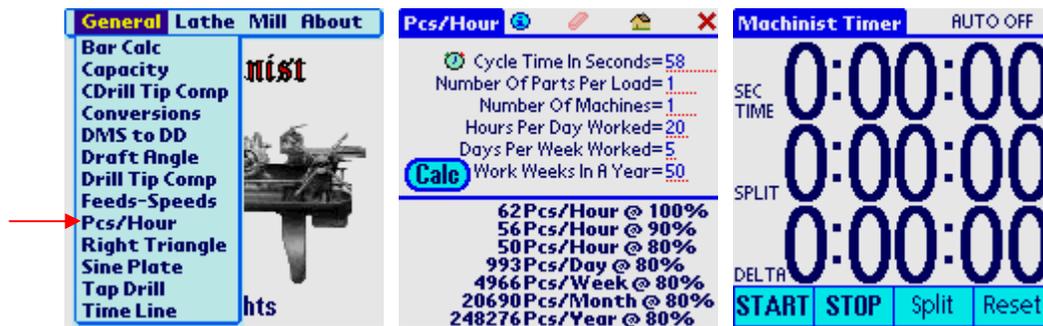
TIP: Only the machine **Cycle Time In Seconds**, the **Number Of Parts Per Load**, and the **Number Of Machines** are required to calculate **Pcs/HR**.

Tap  at any time to access the onboard step by step instructions.

Tap  to clear your information.

Tap  to return to **Machinist** main.

Tap  to exit **Machinist**.



Right Triangle:

This App calculates the unknown sides and angle of a right triangle.

Enter the length of side 'b'.

Enter angle 'C' in decimal degrees.

Tap **Calc** to calculate the remaining angle and sides.

Tap  to clear your information.

Tap  to return to **Machinist** main.

Tap  to exit **Machinist**.



Sine Plate:

This App calculates the thickness of gage blocks needed for any angle on a standard sine plate.

Enter the required Angle.

Enter the sine plate base Length.

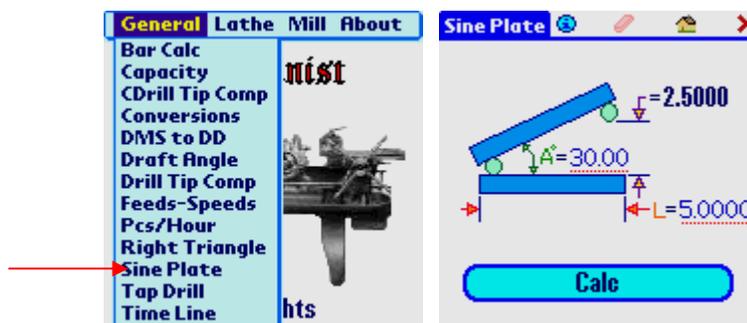
Tap **Calc** to calculate the required gage block thickness.

Tap  at any time to access the onboard step by step instructions.

Tap  to clear your information.

Tap  to return to **Machinist** main.

Tap  to exit **Machinist**.



Tap Drill:

This app allows you to calculate, or look up, drill sizes for tapping.

Enter the 'Thread Diameter'

Enter the 'TPI' (threads per inch) or 'MM Pitch'.

Enter the '% Of Full Thread' (usually 75%).

Tap **Calc** to calculate the Tap Drill Size (in inch or mm).

Tap  to clear your information.

Tap on the down arrow to the right of **Sort Table By**, and select the tap style to sort the data base.

Tap the up/down arrows, or use the Palm hard up/down buttons to navigate the Tap Drill data base.

Read the drill size required.

Tip: Tap the word '**Tap**' in the table header, or tap **Start/Mill/Tap Drill DB** to add/edit the Tap Drill data base.

Tap  at any time to access the onboard step by step instructions.

Tap  to return to **Machinist** main.

Tap  to exit **Machinist**.



Time Line:

This App calculates the total time for project launches, etc.

TIP: Time Line is extremely useful when trying to determine project lead times.

Tap to the right of **Project Start Date**, and select the start date of the project.

Tap to the right of **Project Length (weeks)**, and enter the number of weeks in the project, or tap to the right of **Project Due Date**, and select the due date of your project.

Tap  to calculate the total project launch time.

Tap  at any time to access the onboard step by step instructions.

Tap  to clear your information.

Tap  to return to **Machinist** main.

Tap  to exit **Machinist**.



Grooving:

This App generates EIA program code for rough grooving.

Enter the final -Z- length for the groove.

Enter the groove diameter.

Enter the step over length in the -Z- axis, for multiple grooves.

TIP: Leave the step over blank for single grooves.

Enter the feedrate.

Enter the pecking retract amount.

Enter the pecking depth of cut.

Enter the recess amount at the bottom of the groove.

Tap **Calc** to generate the EIA program code.

Tap  at any time to access the onboard step by step instructions.

Tap  to clear your information.

Tap  to return to **Machinist** main.

Tap  to exit **Machinist**.



Microfinish:

This App calculates the feedrate needed to obtain a required surface finish.

TIP: A micro inch to micro meter, & Ra to Rz conversions are located at the bottom of the page.

Enter the required maximum surface finish.

Enter the insert Tool Nose Radius (TNR).

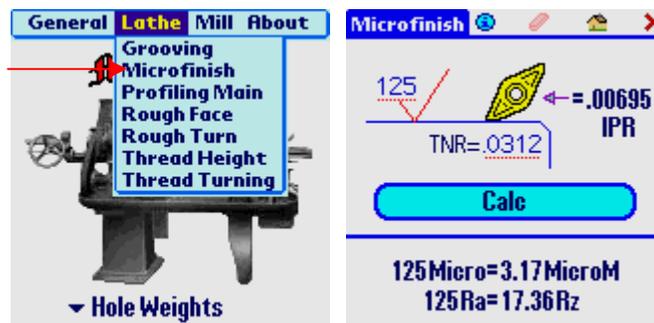
Tap **Calc** to generate the required feedrate.

Tap  at any time to access the onboard step by step instructions.

Tap  to clear your information.

Tap  to return to **Machinist** main.

Tap  to exit **Machinist**.



Profiling Main:

This app generates EIA programming code to simplify radius to angle contouring.

Note: Profiling was created for use on Swiss style CNC lathes, but will work on most CNC lathes with EIA (G-Code) style programming.

Tap on the picture of the profile that you want to program.

Enter the part diameter.

Enter the chamfer length (a positive number).

Enter the ending **Radius**. **Tip:** If no ending **Radius** is required, uncheck the box next to 'R' and leave blank.

Enter the chamfer Angle, in decimal degrees. (**Example:** 15 degrees 30 minutes=15.5)

Enter the start **Radius**. **Tip:** If no starting **Radius** is required, uncheck the box next to 'R' and leave blank.

CAUTION: When no start or end radii are checked, ignore ALL program lines that contain only zeros.

Enter the 'Z' start position.

Enter the **TNR** (Tool Nose **R**adius) on your insert. **Example:** .0156, .0312, .0468, etc.

Tap  for an external (OD) profile, or  for an internal (ID) profile.

Unselect  when using G41/G42. Leave selected for conventional programming.

Tap  if your turret/saddle is on the opposite side of the spindle.

Tap **Calc** to generate the EIA programming code for the selected profile.

Note: Depending on your individual processor speed, it may take up to 30 seconds to finish all of the internal calculations.

Tap  at any time to access the onboard step by step instructions.

Tap  to clear your information.

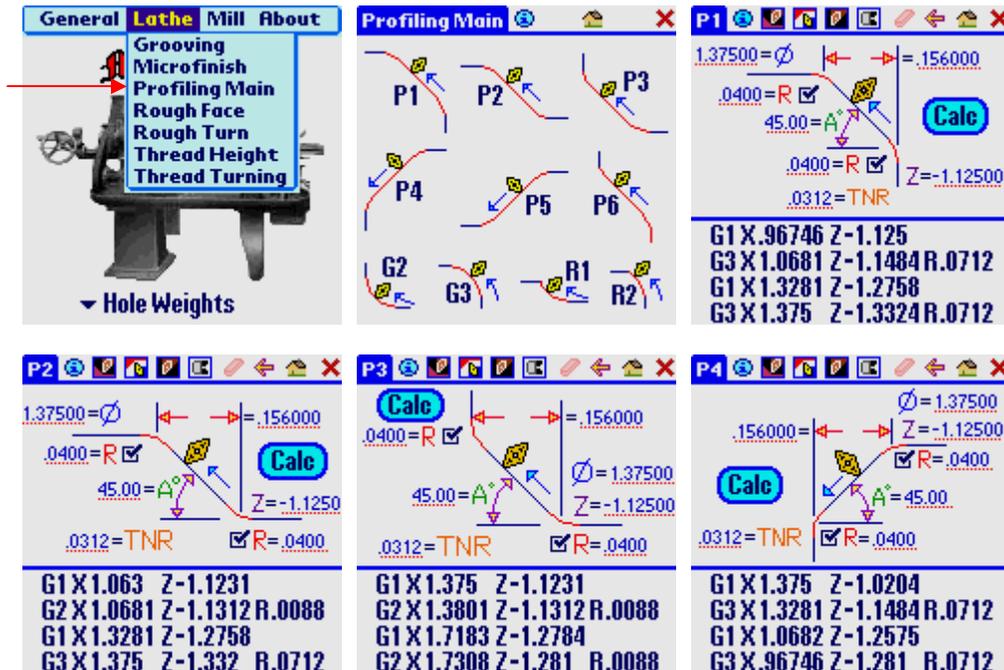
Tap  to return to **Profiling Main**.

Tap  to return to **Machinist** main.

Tap  to exit **Machinist**.

Note: Depending on your individual processor speed, it may take up to 30 seconds to save your information, and exit the app.

CAUTION: These apps are for CNC's that are on centerline within factory specs.



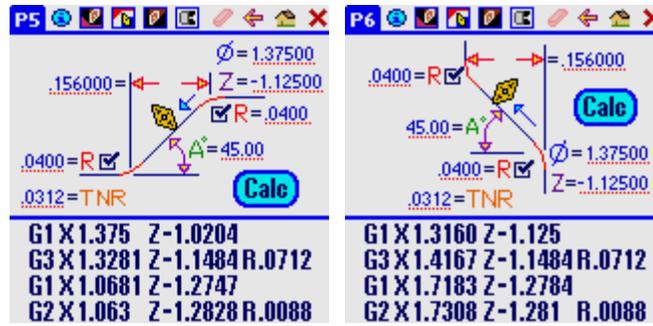
The image displays the Machinist app interface for CNC programming. It shows the 'General' tab with 'Lathe' selected, and a menu of options including Grooving, Microfinish, Profiling Main, Rough Face, Rough Turn, Thread Height, and Thread Turning. The 'Profiling Main' screen shows a diagram of a profile with parameters: P1, P2, P3, P4, P5, P6, G2, G3, R1, R2. The 'Calc' button is highlighted. Below the diagram, the G-code is displayed: G1 X.96746 Z-1.125, G3 X 1.0681 Z-1.1484 R.0712, G1 X 1.3281 Z-1.2758, G3 X 1.375 Z-1.3324 R.0712.

The following G-code is generated for the profile shown in the P1-P4 screens:

```
G1 X 1.063 Z-1.1231
G2 X 1.0681 Z-1.1312 R.0088
G1 X 1.3281 Z-1.2758
G3 X 1.375 Z-1.332 R.0712

G1 X 1.375 Z-1.1231
G2 X 1.3801 Z-1.1312 R.0088
G1 X 1.7183 Z-1.2784
G2 X 1.7308 Z-1.281 R.0088

G1 X 1.375 Z-1.0204
G3 X 1.3281 Z-1.1484 R.0712
G1 X 1.0682 Z-1.2575
G3 X.96746 Z-1.281 R.0712
```



G2 & G3 Radius':

This App generates EIA programming code to simplify radius programming.

Enter the 'Z' axis position.

Enter the **TNR** (Tool Nose **R**adius) on your insert. (Example: .0156, .0312, .0468, etc.)

Enter the required **R**adius.

Enter the diameter.

Tap for external (OD) **R**adii, or for internal (ID) **R**adii.

Unselect when using G41/G42. **TIP:** Leave selected for conventional programming.

Tap if your turret/saddle is on the opposite side of the spindle.

Tap **Calc** to generate the EIA programming code for the selected **R**adius.

Tap at any time to access the onboard step by step instructions.

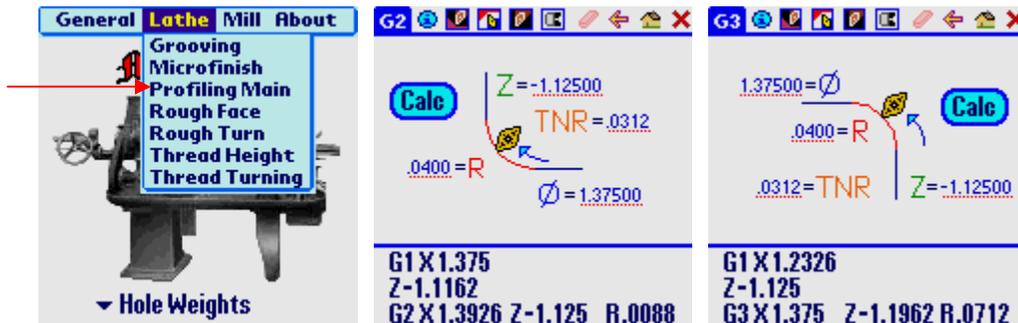
Tap to clear your information.

Tap to return to **Profiling Main**.

Tap to return to **Machinist** main.

Tap to exit **Machinist**.

CAUTION: These apps are for CNC that are on centerline within factory specs.



R1 & R2 Partial Radius':

This App generates EIA programming code to simplify partial radius programming.

Enter the radius ending diameter.

Enter the -Z- axis ending point location.

NOTE: The -Z- axis position must be a negative number, or -0- (i.e. -1.250).

Enter the required **R**adius.

Enter the **TNR** (Tool Nose **R**adius) on your insert. (Example: .0156, .0312, .0468, etc.)

Enter the radius start diameter.

Tap for external (OD) **R**adii, or for internal (ID) **R**adii.

Unselect when using G41/G42. **TIP:** Leave selected for conventional programming.

Tap if your turret/saddle is on the opposite side of the spindle.

Tap **Calc** to generate the EIA programming code.

Tap at any time to access the onboard step by step instructions.

Tap to clear your information.

Tap to return to **Profiling Main**.

Tap to return to **Machinist** main.

Tap to exit **Machinist**.

CAUTION: These apps are for CNC that are on centerline within factory specs.



Rough Face:

This app generates EIA programming code for multiple pass rough facing.

Enter the -Z- axis finish stock allowance.

Enter the -Z- axis roughing depth per pass.

Enter the retract amount.

Enter the -X- axis finish stock allowance.

Enter the block number of the first line of the finish pass of the program (i.e. N100).

Enter the block number of the last line of the finish pass of the program (i.e. N200).

Enter the SFM or MPPM.

Enter the roughing feedrate.

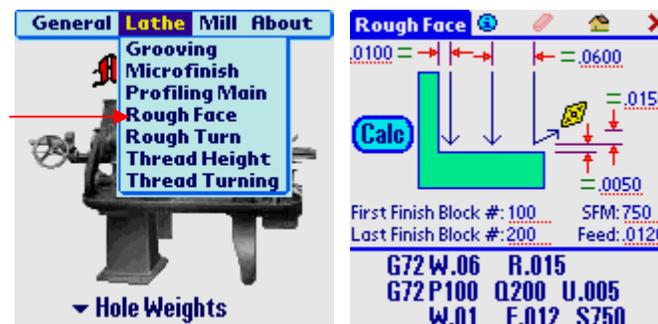
Tap **Calc** to generate the EIA programming code.

Tap at any time to access the onboard step by step instructions.

Tap to clear your information.

Tap to return to **Machinist** main.

Tap to exit **Machinist**.

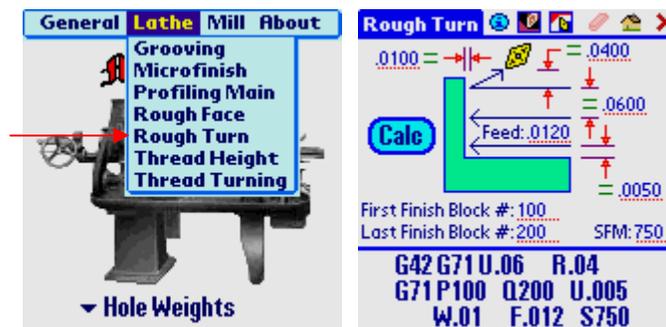


Rough Turn:

This app generates EIA programming code for multiple pass rough turning.

Enter the -Z- axis finish stock allowance.

Enter the retract amount.
 Enter the -X- axis roughing depth per pass.
 Enter the roughing feedrate.
 Enter the -X- axis finish stock allowance.
 Enter the block number of the first line of the finish pass of the program (i.e. N100).
 Enter the block number of the last line of the finish pass of the program (i.e. N200).
 Enter the SFM or MPPM.
 Tap **Calc** to generate the EIA programming code.
 Tap  at any time to access the onboard step by step instructions.
 Tap  to clear your information.
 Tap  to return to **Machinist** main.
 Tap **X** to exit **Machinist**.



Thread Height:

This App calculates internal or external thread heights.
 Enter the **TPI** (Threads Per Inch).
 Tap **Calc** to calculate the **External** and **Internal** thread heights.
 Tap  at any time to access the onboard step by step instructions.
 Tap  to clear your information.
 Tap  to return to **Machinist** main.
 Tap **X** to exit **Machinist**.



Thread Turning:

This App generates EIA program code for multi pass threading.
 Enter the amount of **Taper** in the threads, if needed.
TIP: Leave 'Taper' blank for straight threads.
 Enter the **TPI** (Threads Per Inch).
 Enter the thread diameter.

Enter the **TL** (Total Length of your threads).

Tap **Calc** to generate the EIA programming code.

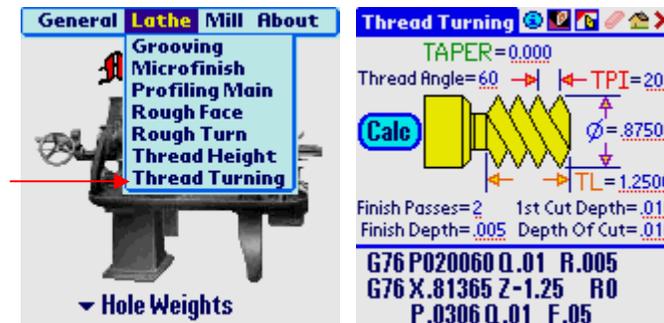
Tap  at any time to access the onboard step by step instructions.

NOTE: Some machines require that you remove the decimal place from the Q, P, & R values. In this case, Q.01 would become Q0100.

Tap  to clear your information.

Tap  to return to **Machinist** main.

Tap  to exit **Machinist**.



Angle Mill:

This app generates EIA programming code for milling around the perimeter of a part.

Enter the cutter diameter.

Enter the length of the first side of the part.

Enter the part angle.

Enter the part width.

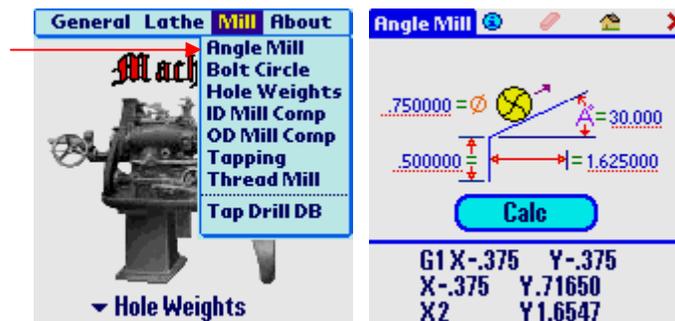
Tap **Calc** to generate the EIA programming code.

Tap  at any time to access the onboard step by step instructions.

Tap  to clear your information.

Tap  again to return to **Machinist** main.

Tap  to exit **Machinist**.



Bolt Circle:

This app generates up to (60) bolt hole locations, regardless of size or start angle position.

Enter the # **HOLES** (number of holes required).

Enter the first hole **START ANGLE** (from the 3 o'clock position).

Enter the **BOLT CIRCLE DIAMETER**.

Enter the 'X' axis bolt pattern center start position.

Enter the 'Y' axis bolt pattern center start position.

Tap **Calc** to generate the EIA programming code for the bolt hole locations.

Note: Depending on your individual processor speed, it may take up to 30 seconds to finish all of the internal calculations.

Tap at any time to access the onboard step by step instructions.

Tap to generate the next set of bolt hole locations, etc.

Tap to view the previous set of bolt hole locations, etc.

Tap to clear your information.

Tap to return to **Bolt Circle** main.

Tap again to return to **Machinist** main.

Tap to exit **Machinist**.

Note: Depending on your individual processor speed, it may take up to 30 seconds to save your information, and exit the app.

General Lathe **Mill** **About**

Angle Mill
Bolt Circle
Hole Weights
ID Mill Comp
OD Mill Comp
Tapping
Thread Mill
Tap Drill DB

▼ Hole Weights

Bolt Circle

HOLES=60
START ANGLE=0.000
BOLT CIRCLE ϕ = 8.75000
-X = 0.00000
-Y = 0.00000 **Calc**

Pos. #	X Axis	Y Axis
Hole Pos. #01=	4.375	0
Hole Pos. #02=	4.35103	.457312
Hole Pos. #03=	4.27939	.909613
Hole Pos. #04=	4.16087	1.35194
Hole Pos. #05=	3.99676	1.77947
Hole Pos. #06=	3.78886	2.18750
Hole Pos. #07=	3.53944	2.57156
Hole Pos. #08=	3.25125	2.92744
Hole Pos. #09=	2.92744	3.25125
Hole Pos. #10=	2.57156	3.53944
Hole Pos. #11=	2.18749	3.78886
Hole Pos. #12=	1.77947	3.99676
Hole Pos. #13=	1.35194	4.16087
Hole Pos. #14=	.909613	4.27939
Hole Pos. #15=	.457311	4.35103

Pos. #	X Axis	Y Axis
Hole Pos. #16=	-0.00000	4.375
Hole Pos. #17=	-.45731	4.35103
Hole Pos. #18=	-.90961	4.27939
Hole Pos. #19=	-1.35194	4.16087
Hole Pos. #20=	-1.77947	3.99676
Hole Pos. #21=	-2.18750	3.78886
Hole Pos. #22=	-2.57156	3.53944
Hole Pos. #23=	-2.92744	3.25125
Hole Pos. #24=	-3.25125	2.92744
Hole Pos. #25=	-3.53944	2.57156
Hole Pos. #26=	-3.78886	2.18749
Hole Pos. #27=	-3.99676	1.77947
Hole Pos. #28=	-4.16087	1.35194
Hole Pos. #29=	-4.27939	.909613
Hole Pos. #30=	-4.35110	.457311

Pos. #	X Axis	Y Axis
Hole Pos. #31=	-4.375	-0.00000
Hole Pos. #32=	-4.3510	-.45731
Hole Pos. #33=	-4.2793	-.90961
Hole Pos. #34=	-4.1608	-1.3519
Hole Pos. #35=	-3.9967	-1.7794
Hole Pos. #36=	-3.7888	-2.1875
Hole Pos. #37=	-3.5394	-2.5715
Hole Pos. #38=	-3.2512	-2.9274
Hole Pos. #39=	-2.9274	-3.2512
Hole Pos. #40=	-2.5715	-3.5394
Hole Pos. #41=	-2.1874	-3.7888
Hole Pos. #42=	-1.7794	-3.9967
Hole Pos. #43=	-1.3519	-4.1608
Hole Pos. #44=	-.90961	-4.2793
Hole Pos. #45=	-.45731	-4.3510

Pos. #	X Axis	Y Axis
Hole Pos. #46=	0.00000	-4.375
Hole Pos. #47=	.457312	-4.3510
Hole Pos. #48=	.909614	-4.2793
Hole Pos. #49=	1.35194	-4.1608
Hole Pos. #50=	1.77947	-3.9967
Hole Pos. #51=	2.18750	-3.7888
Hole Pos. #52=	2.57156	-3.5394
Hole Pos. #53=	2.92744	-3.2512
Hole Pos. #54=	3.25125	-2.9274
Hole Pos. #55=	3.53944	-2.5715
Hole Pos. #56=	3.78886	-2.1874
Hole Pos. #57=	3.99676	-1.7794
Hole Pos. #58=	4.16087	-1.3519
Hole Pos. #59=	4.27939	-.90961
Hole Pos. #60=	4.35103	-.45731

Hole Weights:

This app calculates the weight of the material removed from drilling a hole at a given location, for balancing purposes.

Enter the drill depth.

Enter the drill diameter.

Enter the radial location of the drilled hole.

Select the type of material being drilled

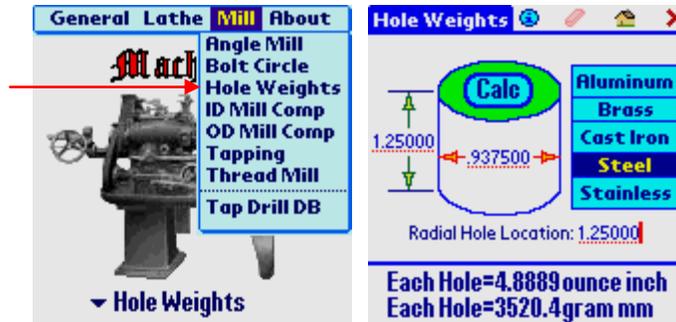
Tap **Calc** to calculate the drilled material weights.

Tap at any time to access the onboard step by step instructions.

Tap to clear your information.

Tap to return to **Machinist** main.

Tap to exit **Machinist**.



ID Mill Comp:

This app generates EIA programming code for pocket milling.

Enter the diameter of the tool being used.

Enter the bore diameter to be milled.

Tap for a clockwise cutter path.

Tap for a counter clockwise cutter path.

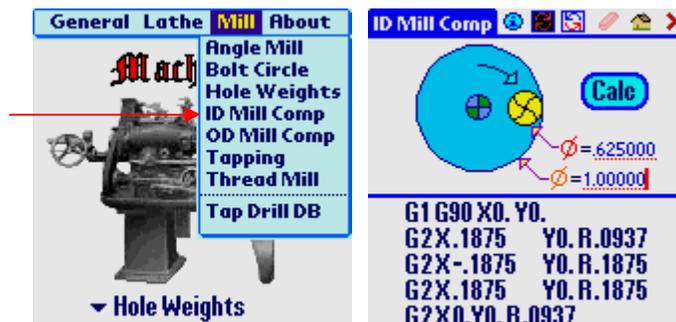
Tap **Calc** to generate the EIA programming code.

Tap at any time to access the onboard step by step instructions.

Tap to clear your information.

Tap to return to **Machinist** main.

Tap to exit **Machinist**.



OD Mill Comp:

This app generates EIA programming code for boss milling.

Enter the diameter of the tool being used.

Enter the required boss diameter to be milled.

Tap for a clockwise cutter path.

Tap for a counter clockwise cutter path.

Tap **Calc** to generate the EIA programming code.

Tap at any time to access the onboard step by step instructions.

Tap to clear your information.

Tap to return to **Machinist** main.

Tap to exit **Machinist**.



Tapping:

This App generates EIA programming code for tapping in a machining center.

Enter the 'X' axis tap location.

Enter the 'Y' axis tap location.

Enter the desired **RPM** (Revolutions Per Minute).

Enter the **TPI** (Threads Per Inch).

Enter the **TL** (Total Length of your threads).

Tap **Calc** to generate the EIA programming code.

Tap at any time to access the onboard step by step instructions.

Tap to clear your information.

Tap to return to **Machinist** main.

Tap to exit **Machinist**.



Thread Mill:

This App is used to ease EIA thread mill programming.

Tap for external (OD) threads, or for internal (ID) threads.

Enter the length or depth of thread (**Thread Depth**).

Enter the diameter of the thread (**Thread Diameter**).

Enter the 'Feed' per tooth (i.e. .003).

Enter the diameter of the thread mill.

Enter the recommended **SFM** (Surface Feet per Minute) for your material.

Enter The **TPI** (Threads Per Inch).

Tap **Calc** to generate the EIA programming code.

Tap at any time to access the onboard step by step instructions.

Tap to clear your information.

Tap to return to **Machinist** main.

Tap to exit **Machinist**.



Tap Drill DB:

This App is used to enter all of your taps drill sizes into the data base.

Tap to the right of 'Select Tap Type' and select the category for the tap being entered.

Enter the 'Tap' information.

Enter the 'Tap Drill' information.

Enter the 'Decimal Equivalent' of the tap drill.

Tap \leftarrow to view/edit the first tap in the data base.

Tap \leftarrow to view/edit the previous tap in the data base.

Tap \pencil to add a new tap to the data base.

Tap \rightarrow to view/edit the next tap in the data base.

Tap \rightarrow to view/edit the last tap in the data base.

Tap \odot at any time to access the onboard step by step instructions.

Tap eraser to clear your information.

Tap trash to delete a tap from the data base.

Note: This action CAN NOT be undone.

Tap \leftarrow to go to Tap Drill main.

Tap home to return to **Machinist** main.

Tap \times to exit **Machinist**.



EULA (End User License Agreement):

Palm is a registered trademark of **Palm One**.

You may install and use only **one** (1) copy of **Machinist** on any one device at any time.

You may **not** reproduce or distribute registered copies of **Machinist**.

Machinist may **not** be disassembled, decompiled, or reverse engineered.

The developer cannot be held responsible for losses incurred from this application.

This application is provided “as is” without any implied warranty.

Installation of this application automatically implies consent to the **previous** statements.

Arzdog Software reserves the right to change this application and documentation at any time.

This EULA may be terminated if you fail to comply with the terms and conditions herein.

Please forward any comments, suggestions, questions, or bugs to arzdog@msn.com.