

# Machinist User Guide



# Table Of Contents

## Introduction:

Preface	page 3
---------	--------

## Turning:

1) Turning Main	page 4
2) G2 & G3 Radius'	page 6
3) Partial Face & Partial Turn Radius'	page 8
4) Rough Face	page 10
5) Rough Turn	page 11
6) Microfinish	page 12
7) Threading	page 13
8) Thread Height	page 14
9) Grooving	page 15

## Milling:

11) Bolt Circle	page 16
12) Hole Weight	page 17
13) Tapping	page 18
14) Angle Mill	page 19
15) OD Mill	page 20
16) ID Mill	page 21
17) Thread Milling	page 22
18) Center Drill Tip Comp.	page 23
19) Drill Tip Comp.	page 24
20) Tap Drill Selector	page 25
21) Tap Drill Data Base	page 26

## Communications:

22) DNC	page 27
---------	---------

## Formulas:

23) Bar Calc	page 29
24) DMS to Dec. Deg.	page 30
25) Project Time Line	page 31
26) Capacity	page 32
27) Draft Angle	page 33
28) Sine Plate	page 34
29) Unit Conversion	page 35
30) Pcs / Hour	page 36
31) Feeds-Speeds	page 37
32) Trig Calc	page 38

## Miscellaneous:

33) EULA (End User License Agreement)	page 39
---------------------------------------	---------

## Preface:

**Machinist** is **THE** premier application for engineers, CNC programmers, set-up, toolmakers, or anyone in the machining industry. **Machinist** is designed to ease CNC lathe programming of complex profile contouring. **Machinist** generates up to (40) bolt hole locations, regardless of start angle or position. **Machinist** simplifies drill tip depth compensation. **Machinist** generates CNC threading cycles for turning, tapping, and thread milling. **Machinist** also calculates process cycle times and efficiency, plus much, much more. The highly graphical interface is user friendly and very intuitive.

**Minimum OS requirement:** Any device running PocketPC 2002 and newer, including Windows Mobile 6.

**Read the [EULA](#) section at the end of this User Guide before proceeding.**

Read the READ ME FIRST.txt document before installing **Machinist**.

Unzip the install file.

Open the unzipped file and navigate to the PocketPC folder.

Double click the setup.exe file to start the install.


When the install is completed, copy the Material.txt & TapDrillDb.txt files into the **Machinist** folder located in the Program Files folder on the handheld.


Follow the instructions in the READ ME FIRST.txt file for importing the data into **Machinist**.

The Demo version of **Machinist** is fully functional for 15 uses.

After the Demo has expired, a registered version of **Machinist** must be purchased to continue unrestricted usage.

Tap Start/Programs/**Machinist** to start the application.

Tap  , located at the top left of every page, at any time to access the extensive onboard user manual.

Tap  , located at the top right of every page, to exit **Machinist**.

## Turning Main:

This app was created for use on Swiss style CNC lathes, but will work on most modern CNC lathes with EIA (G-Code) style programming.

Tap the lathe short cut on the main page, or select from the drop down menu.

Tap the P1 thru P6 short cut for the profile that you want to program.

Select OD or ID machining.

Leave Comp selected for standard programming. Unselect if you are using G41/G42.

Select Opposite if your turret/saddle is on the opposite side of the spindle (left of the spindle).

Enter the part diameter.

Enter the chamfer length (must be a positive number).

Enter the ending radius, if required.


Enter the chamfer angle, in decimal degrees. (example: 15 degrees 30 minutes=15.5)


Enter the start corner radius, if required.


Enter the 'Z' axis start position (Must be 0 or a negative number).

Enter the TNR (tool nose radius) on your insert. (example: .0156, .0312, .0468, etc).

Tap  to access the onboard step by step instructions.

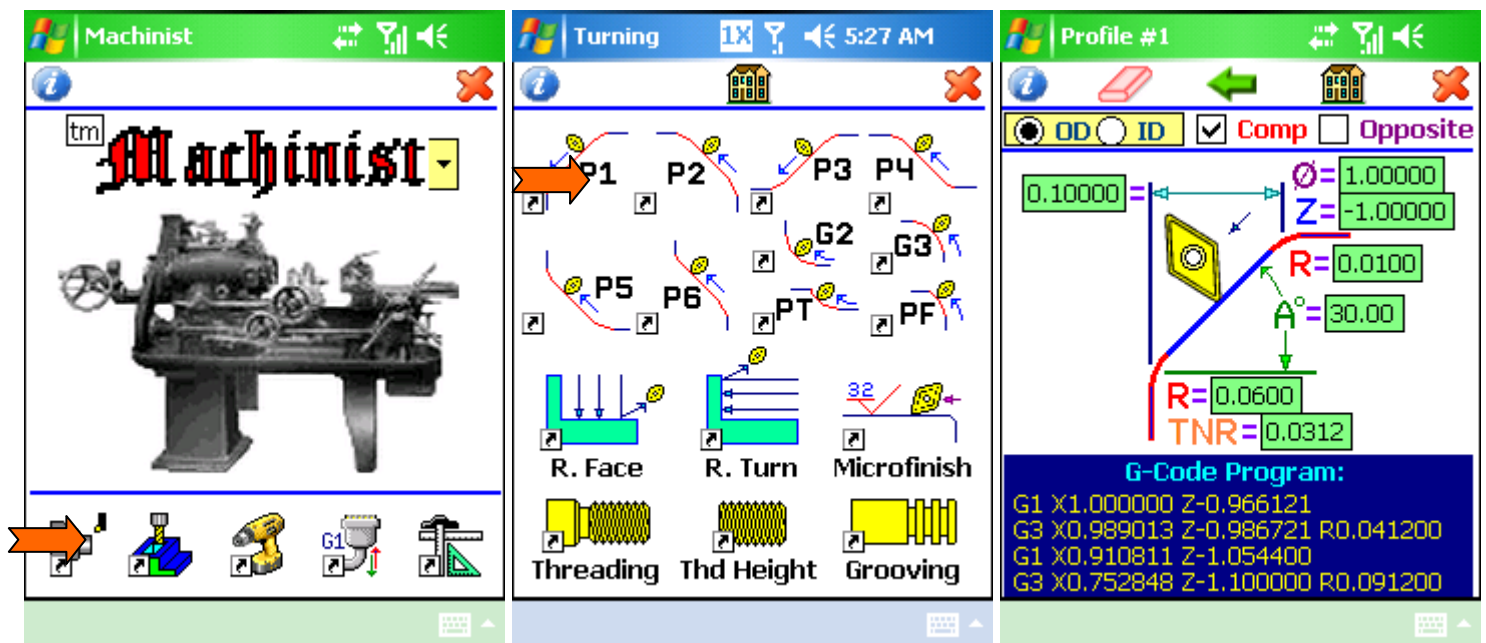
Tap  to clear your information.

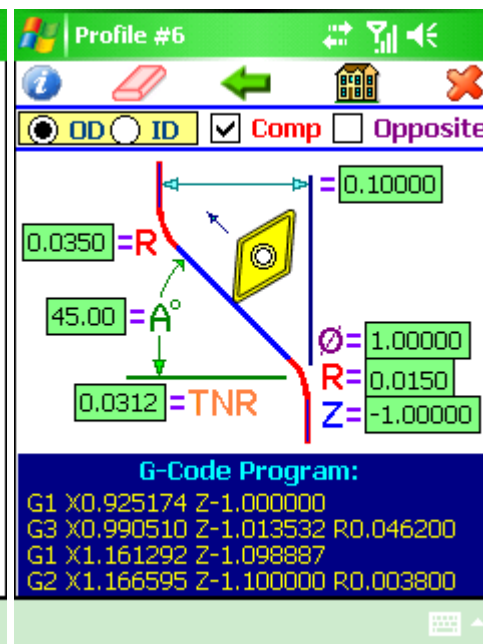
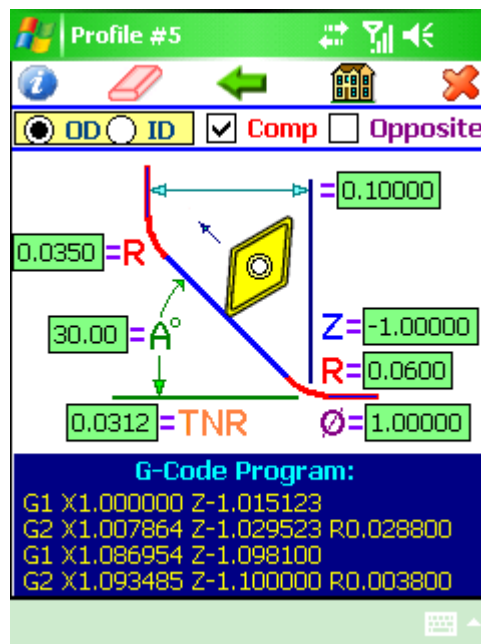
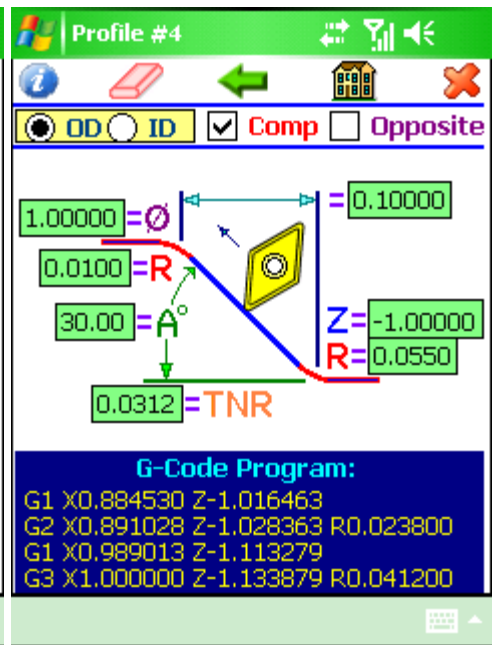
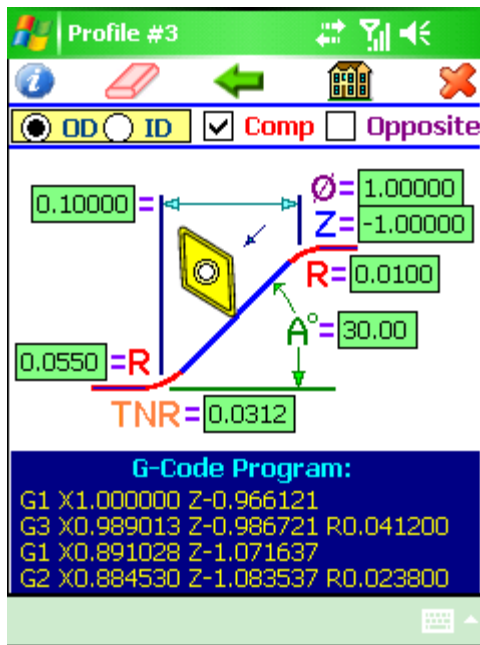
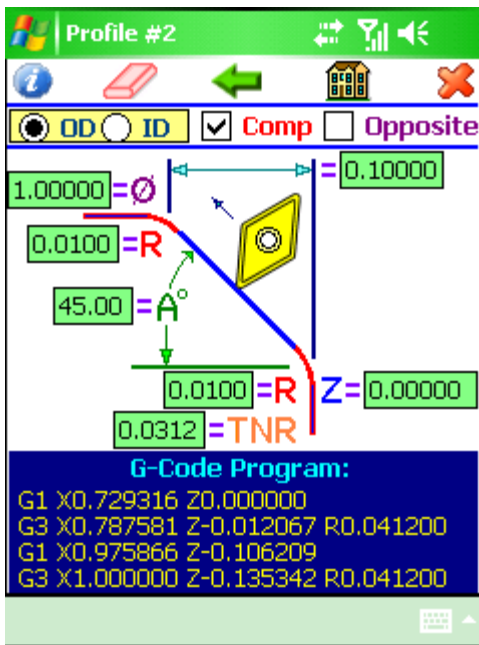
Tap  to return to Turning main.

Tap  to return to **Machinist** main.

Tap  to exit **Machinist**.

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





## G2 & G3 Radii:

These apps generate EIA programming code to simplify radius programming.

Tap the lathe short cut on the main page, or select from the drop down menu.

Tap the G2 (clockwise) or G3 (counter clockwise) radius.

Select OD or ID machining.

Leave Comp selected for standard programming. Unselect if you are using G41/G42.


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


Enter the 'Z' axis position (must be 0 or a negative number).


Enter the TNR (tool nose radius) on your insert. (example: .0156, .0312, .0468, etc.)


Enter the required radius.

Enter the start or end diameter.

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

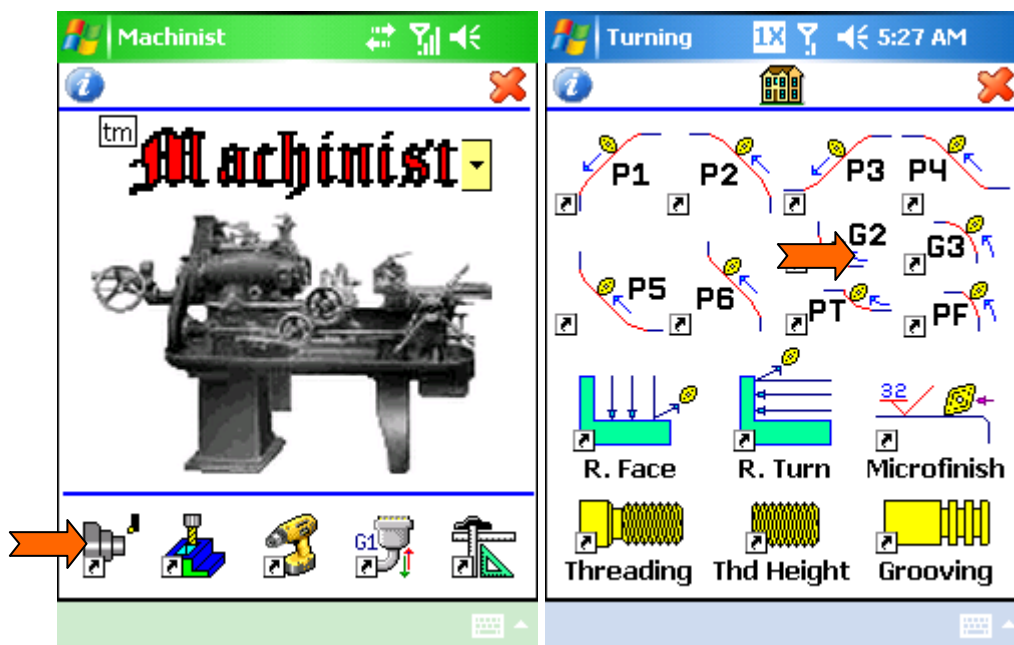
Tap  to clear your information.

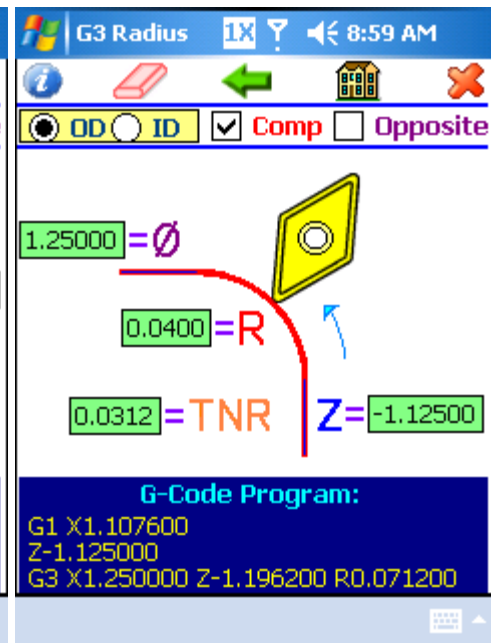
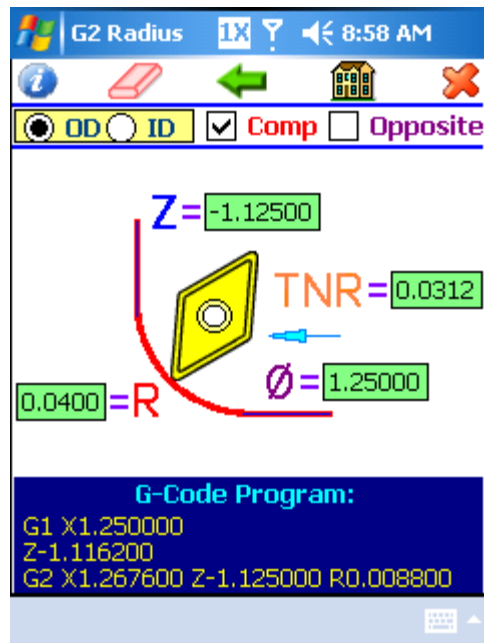
Tap  to return to Turning main.

Tap  to return to **Machinist** main.

Tap  to exit **Machinist**.

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





## Partial Face & Partial Turn Radius:

These apps generate EIA programming code to simplify partial radius programming.

Tap the lathe short cut on the main page, or select from the drop down menu.

Tap the PT (partial turn) or PF (partial face) short cut.

Select OD or ID machining.

Leave Comp selected for standard programming. Unselect if you are using G41/G42.

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


Enter the radius ending diameter.


Enter the -Z- axis ending point location (must be a 0 or negative number).


Enter the required radius.


Enter the TNR (tool nose radius) on your insert. (example: .0156, .0312, .0468, etc.)


Enter the radius start diameter.

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

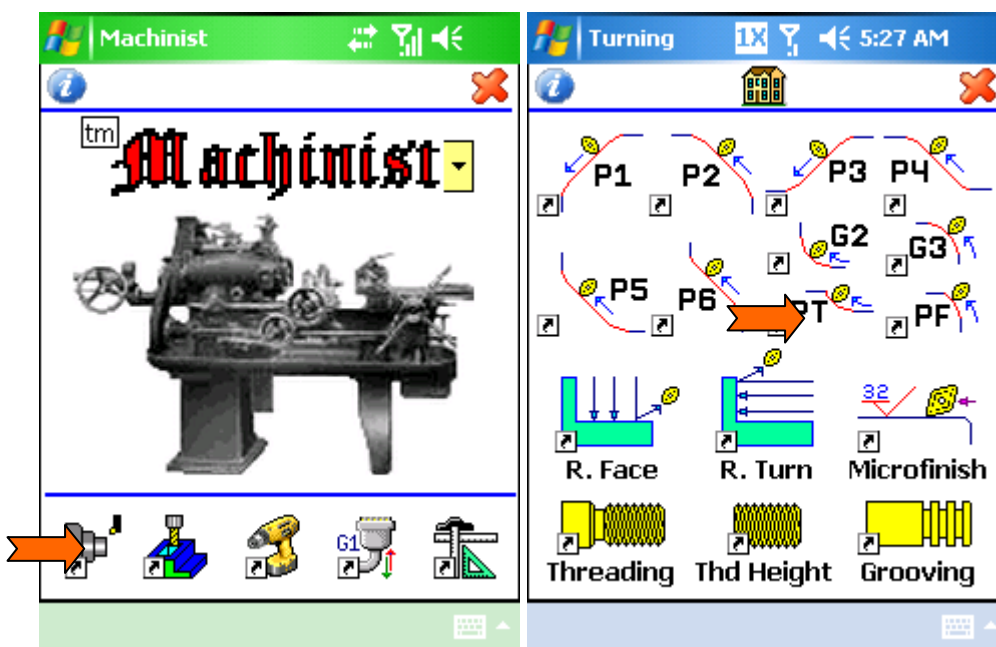
Tap  to clear your information.

Tap  to return to Turning main.

Tap  to return to **Machinist** main.

Tap  to exit **Machinist**.

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





Partial Turn 1X 9:46 AM

OD ID ☒ Comp ☐ Opposite

1.37500 = Ø  
-1.12500 = Z  
0.5000 = R Ø = 1.25000  
TNR = 0.0312

**G-Code Program:**

```
G1 X1.250000
Z-0.882939
G2 X1.375000 Z-1.125000 R0.468800
```

Partial Face 1X 9:48 AM

OD ID ☒ Comp ☐ Opposite

0.0312 = TNR  
1.25000 = Ø  
0.5000 = R  
1.00000 = Ø Z = -1.12500

**G-Code Program:**

```
G1 Z-1.125000
X1.000000
G3 X1.250000 Z-1.455719 R0.531200
```

## Rough Face:

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

Tap the lathe short cut on the main page, or select from the drop down menu.

Tap the R. Face short cut.

Leave Comp selected for standard programming. Unselect if you are using G41/G42.

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

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 (surface feet per minute).

Enter the roughing feedrate.

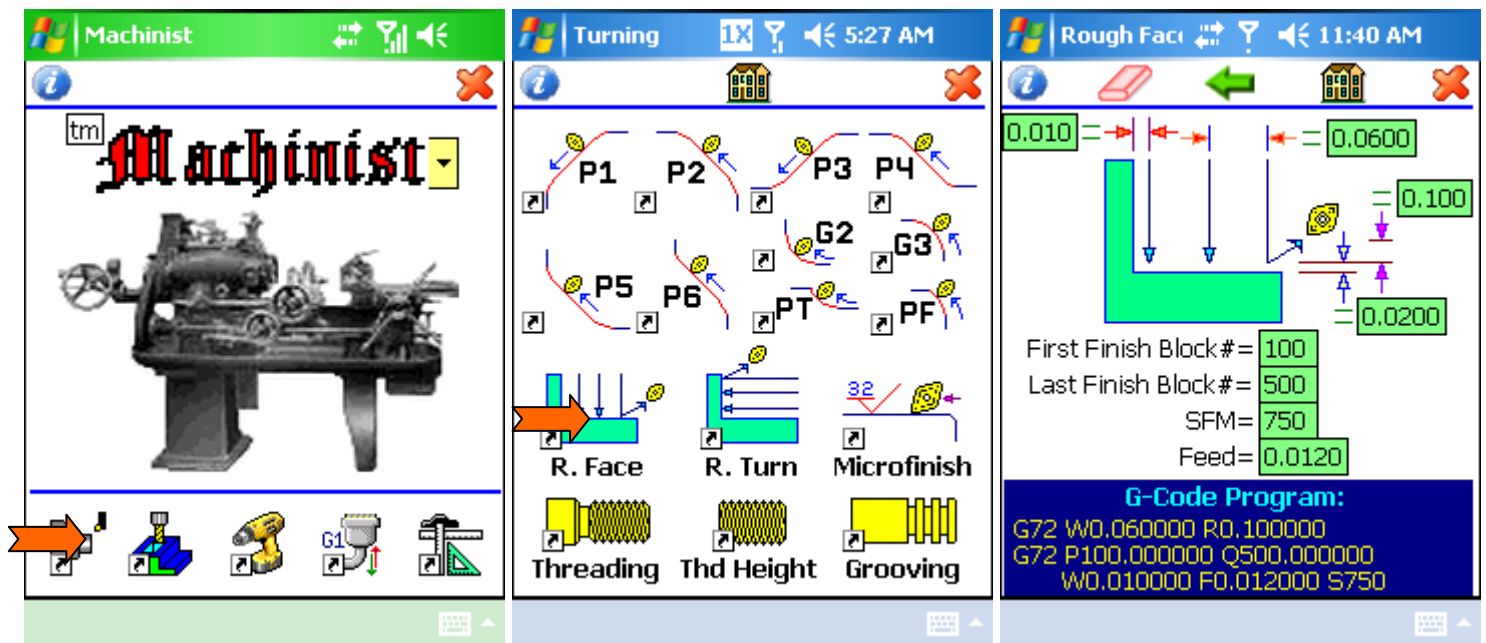
Tap  at any time to access the onboard step by step instructions.

Tap  to clear your information.






Tap  to return to Turning main.

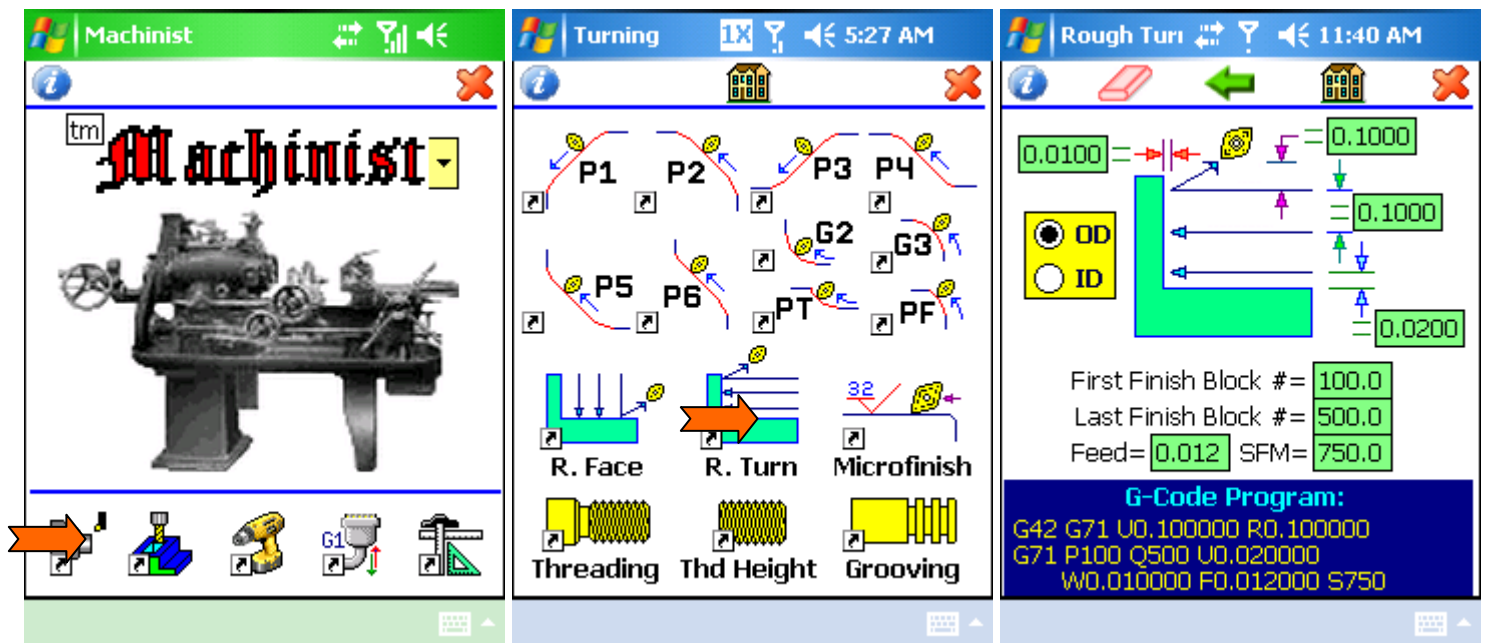
Tap  to return to **Machinist** main.

Tap  to exit **Machinist**.



## Rough Turn:

- This app generates EIA programming code for multiple pass rough turning.
- Tap the lathe short cut on the main page, or select from the drop down menu.
- Tap the R.Turn short cut.
- Select OD or ID machining.
- Leave Comp selected for standard programming. Unselect if using G41/G42.
- Tap Opposite if your turret/saddle is on the opposite side of the spindle.
- Enter the -Z- axis finish stock allowance.
- Enter the retract amount.
- Enter the -X- axis roughing depth per pass.
- 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 roughing feedrate.
- Enter the SFM (surface feet per minute).
- Tap  at any time to access the onboard step by step instructions.
- Tap  to clear your information.
- Tap  to return to Turning main.
- Tap  to return to **Machinist** main.
- Tap  to exit **Machinist**.



## Microfinish:

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


Tap the lathe short cut on the main page, or select from the drop down menu.


Tap the Microfinish short cut.


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

Enter the required maximum surface finish.

Enter the insert TNR (tool nose radius).

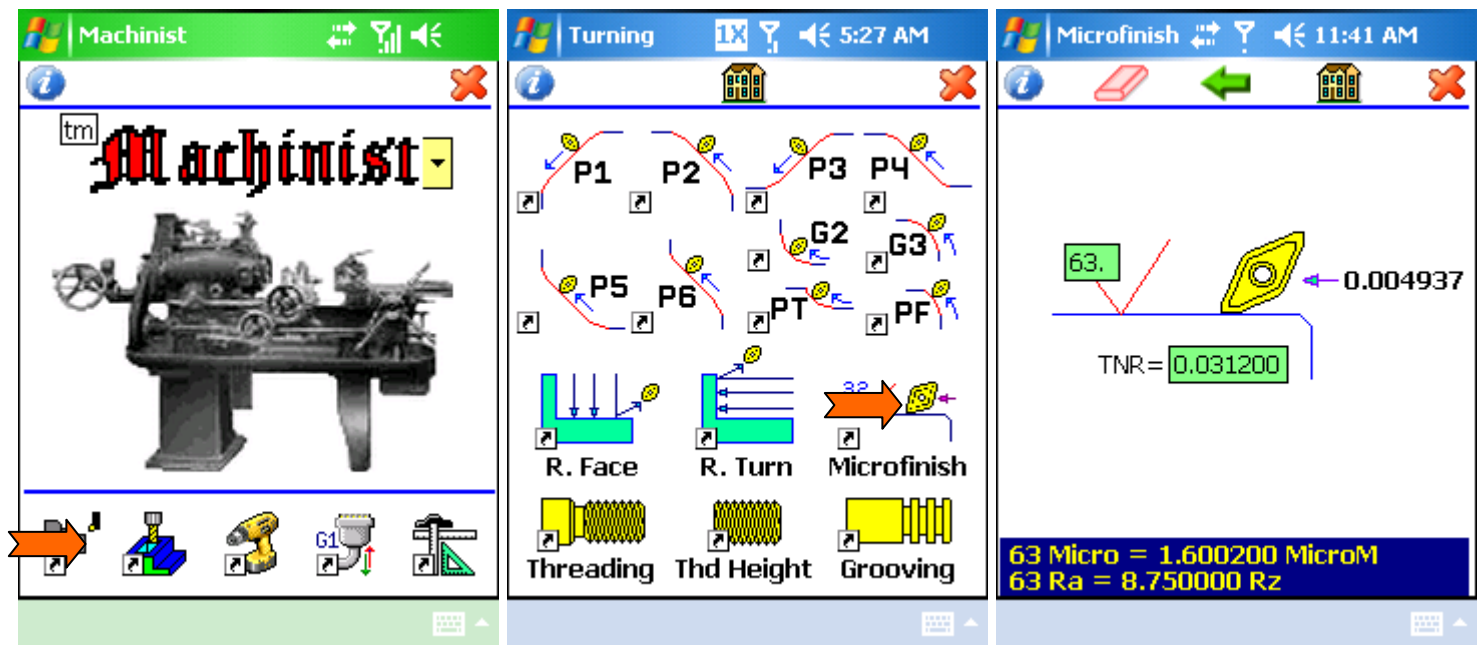
Tap  at any time to access the onboard step by step instructions.

Tap  to clear your information.

Tap  to return to Turning main.

Tap  to return to **Machinist** main.

Tap  to exit **Machinist**.



## Threading:

This app generates EIA program code for multi pass threading.

Tap the lathe short cut on the main page, or select from the drop down menu.

Tap the Threading short cut.

Select OD or ID machining.

Enter the TPI (threads per inch).

Enter the thread diameter.

Enter the total length of your threads.

Enter the amount of taper in the threads, if required.

**TIP:** Leave taper blank for straight threads.

Enter the thread angle (normally 60 degrees).


Enter the first thread pass depth of cut.


Enter the decremental (standard) thread pass depth of cut.


Enter the number of finish thread passes.


Enter the finish pass depth of cut.

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

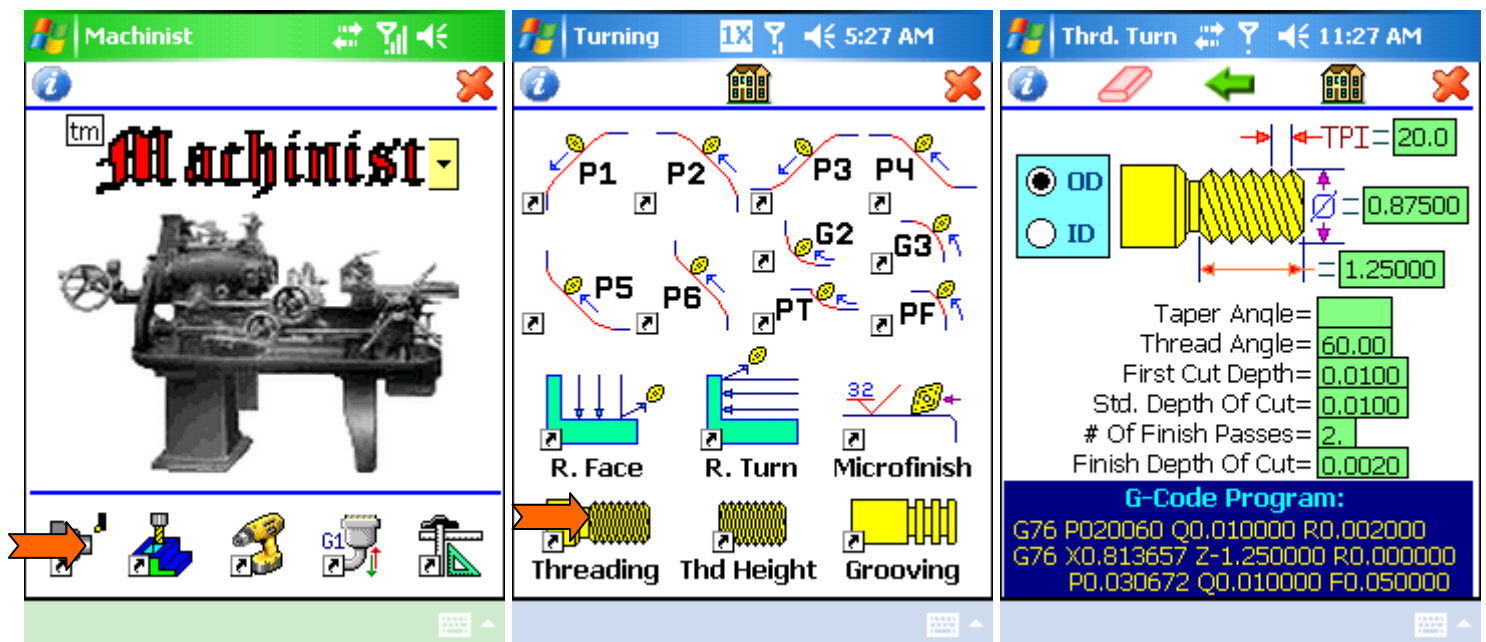
Tap  at any time to access the onboard step by step instructions.

Tap  to clear your information.

Tap  to return to Turning main.

Tap  to return to **Machinist** main.

Tap  to exit **Machinist**.



## Thread Height:


This app calculates internal or external thread heights.


Tap the lathe short cut on the main page, or select from the drop down menu.


Tap the Thd Height short cut.


Select OD or ID machining.

Enter the TPI (threads per inch).

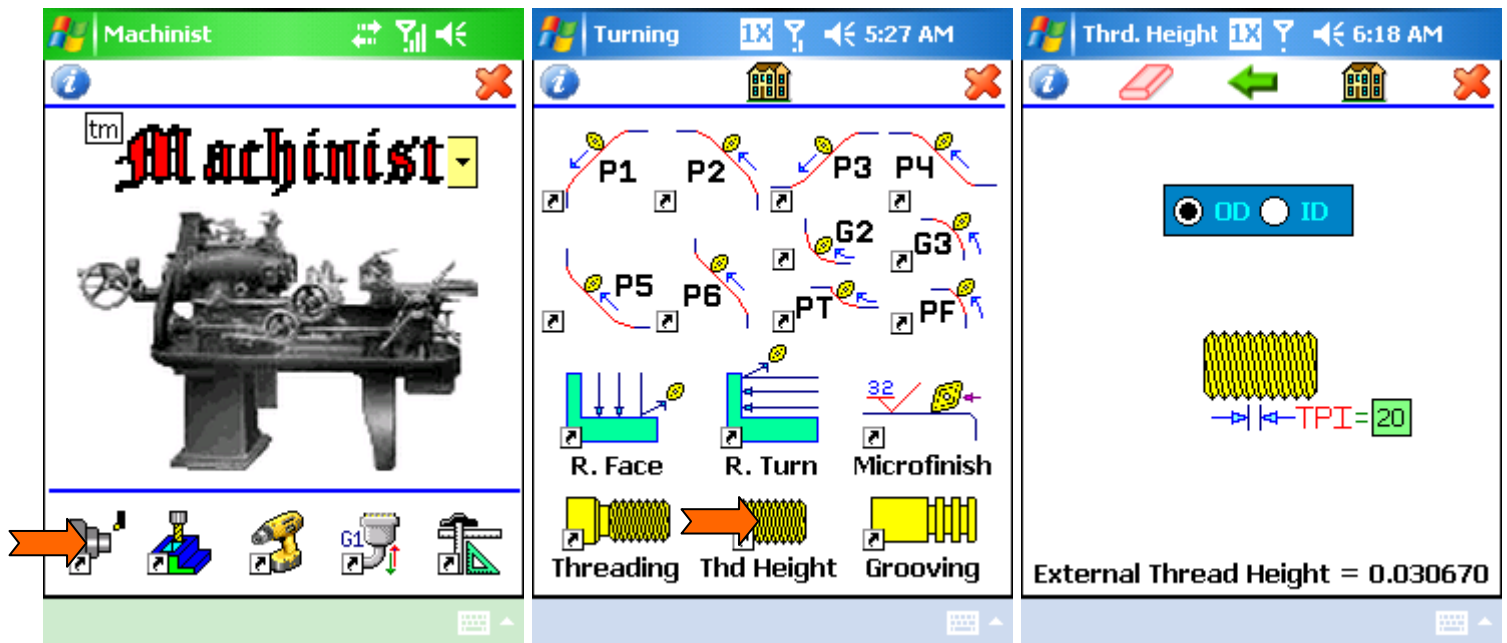
Tap  at any time to access the onboard step by step instructions.

Tap  to clear your information.

Tap  to return to Turning main.

Tap  to return to **Machinist** main.

Tap  to exit **Machinist**.



## Grooving:

This app generates EIA program code for rough grooving.

Tap the lathe short cut on the main page, or select from the drop down menu.

Tap the Grooving short cut.

Enter the final –Z- axis position of the last groove.

Enter the groove diameter.

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


**TIP:** Leave the step over blank for a single groove.


Enter the feedrate.


Enter the pecking retract amount.


Enter the pecking depth of cut.


Enter the recess amount at the bottom of the groove.

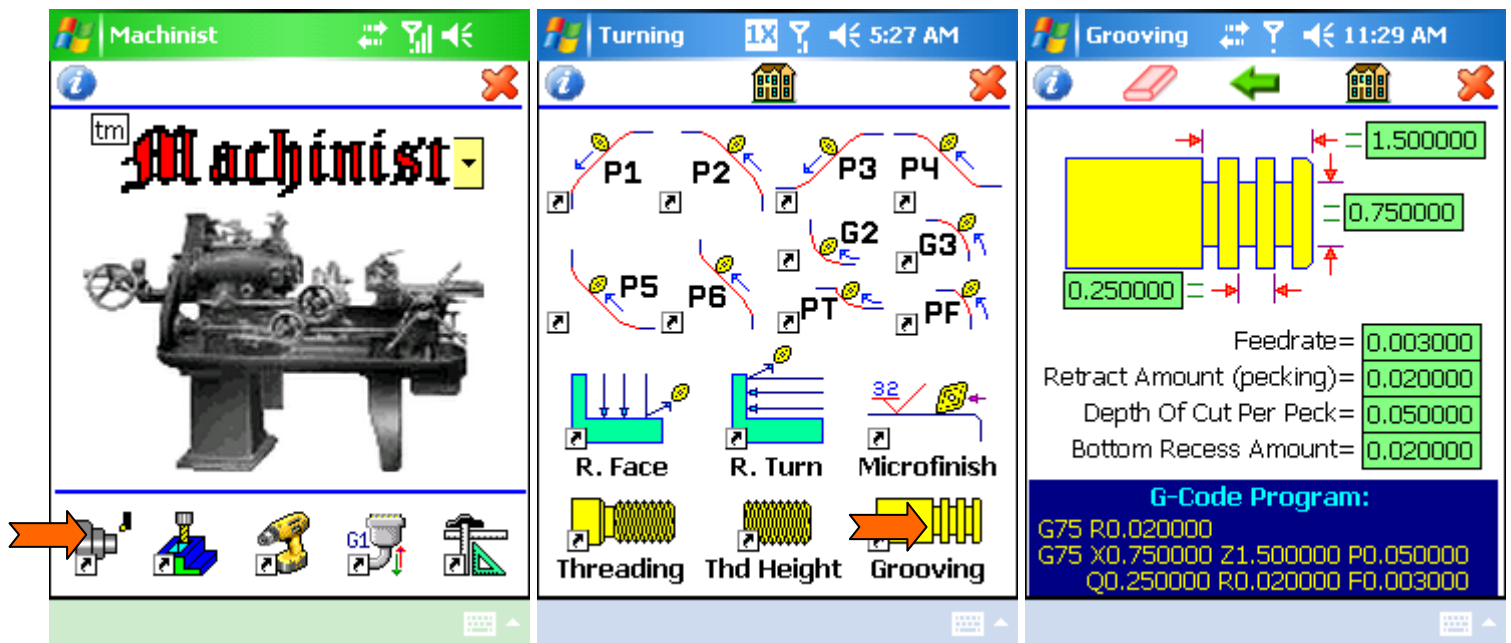
Tap  at any time to access the onboard step by step instructions.

Tap  to clear your information.

Tap  to return to Turning main.

Tap  to return to **Machinist** main.

Tap  to exit **Machinist**.



## Bolt Circle:

This app generates up to (40) bolt hole locations, regardless of size or start angle position.

Tap the milling short cut on the main page, or select from the drop down menu.

Tap the Bolt Circle short cut.

Enter the # of holes required.


Enter the first hole start angle (from the 3 o'clock position).


Enter the 'X' axis bolt pattern center position.


Enter the 'Y' axis bolt pattern center position.


Enter the bolt circle diameter.


Use the right scroll bar to view you bolt hole locations.

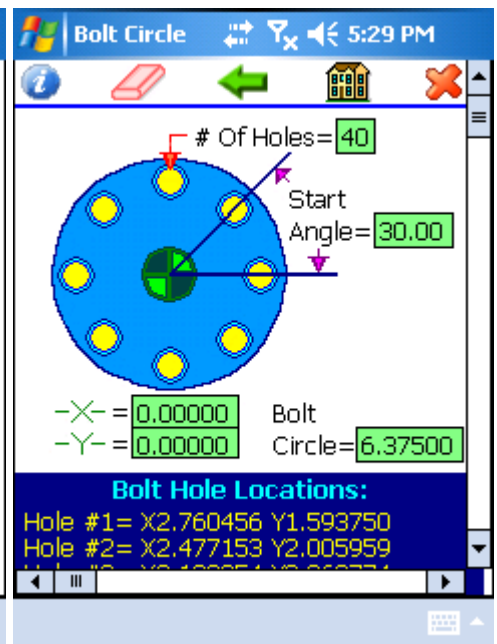
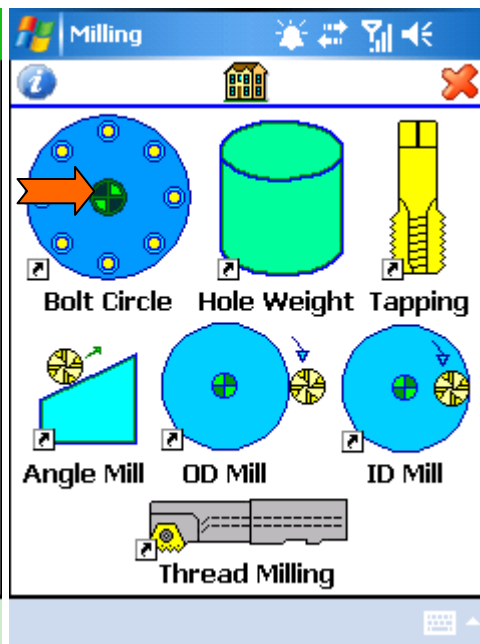
Tap  at any time to access the onboard step by step instructions.

Tap  to clear your information.

Tap  to return to Milling main.

Tap  to return to **Machinist** main.

Tap  to exit **Machinist**.





## Hole Weight:

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

Tap the milling short cut on the main page, or select from the drop down menu.


Tap the Hole Weight short cut.


Enter the drill depth.


Enter the drill diameter.


Enter the radial location of the drilled hole.

Select the type of material being drilled.

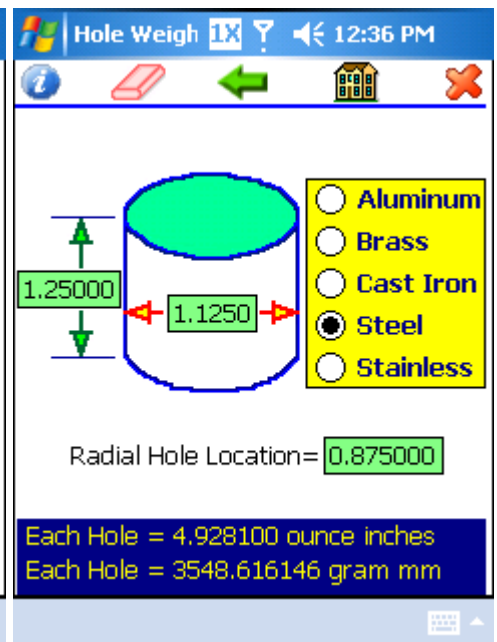
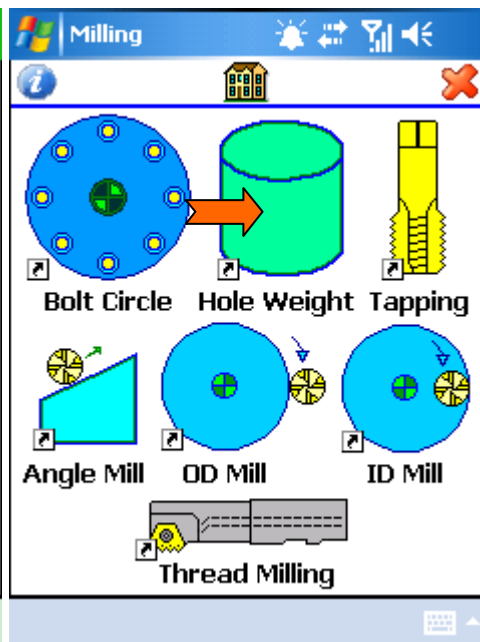
Tap  at any time to access the onboard step by step instructions.

Tap  to clear your information.






Tap  to return to Milling main.

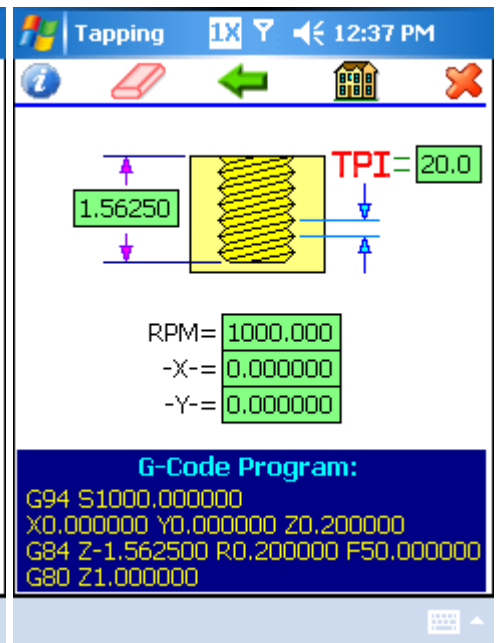
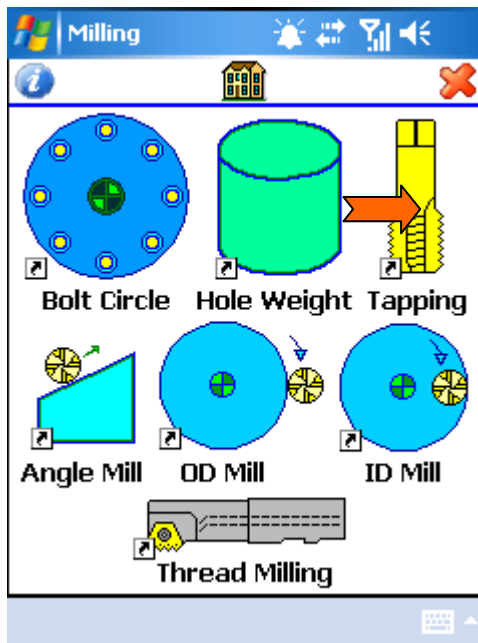
Tap  to return to **Machinist** main.

Tap  to exit **Machinist**.



## Tapping:

- This app generates EIA programming code for tapping in a machining center.
- Tap the milling short cut on the main page, or select from the drop down menu.
- Tap the Tapping short cut.
- Enter the total thread depth.
- Enter the TPI (threads per inch).
- Enter the desired RPM (revolutions per minute).
- Enter the 'X' axis tap location.
- Enter the 'Y' axis tap location.
- Tap  at any time to access the onboard step by step instructions.
- Tap  to clear your information.
- Tap  to return to Milling main.
- 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.

Tap the milling short cut on the main page, or select from the drop down menu.


Tap the Angle Mill short cut.


Enter the cutter diameter.


Enter the length of the first milled side of the part.


Enter the angle to be milled.

Enter the part milled width.

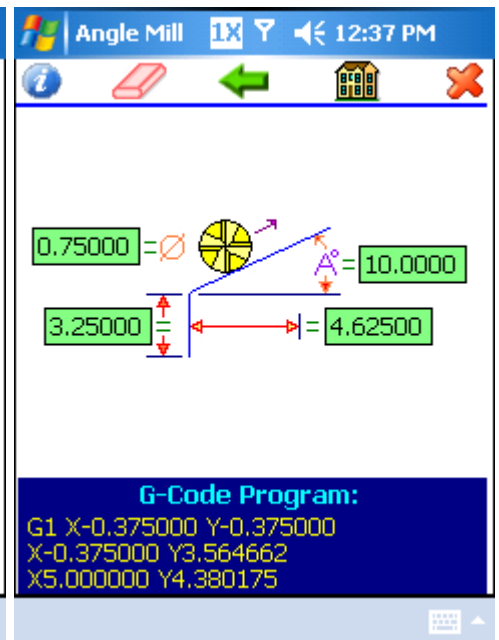
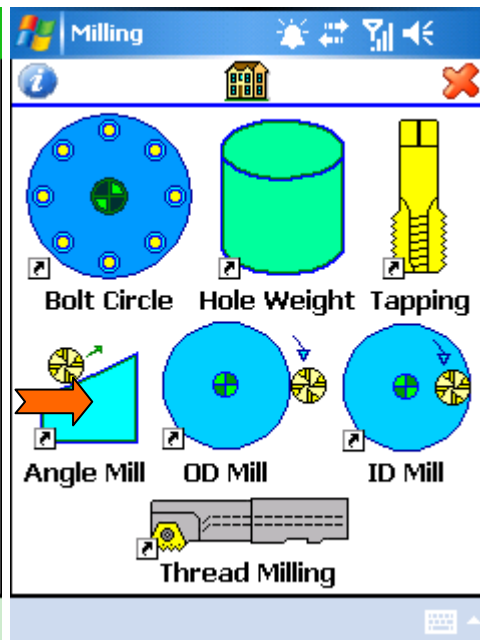
Tap  at any time to access the onboard step by step instructions.

Tap  to clear your information.

Tap  to return to Milling main.

Tap  to return to **Machinist** main.

Tap  to exit **Machinist**.



## O.D. Mill:

This app generates EIA programming code for boss milling.

Tap the milling shortcut on the main page, or select from the drop down menu.

Tap the OD Mill short cut.


Select CW (clockwise) or CCW (counter clockwise) cutter path.


Enter the diameter of the tool being used.


Enter the required boss diameter to be milled.


Enter the -X- axis boss location.

Enter the -Y- axis boss location.

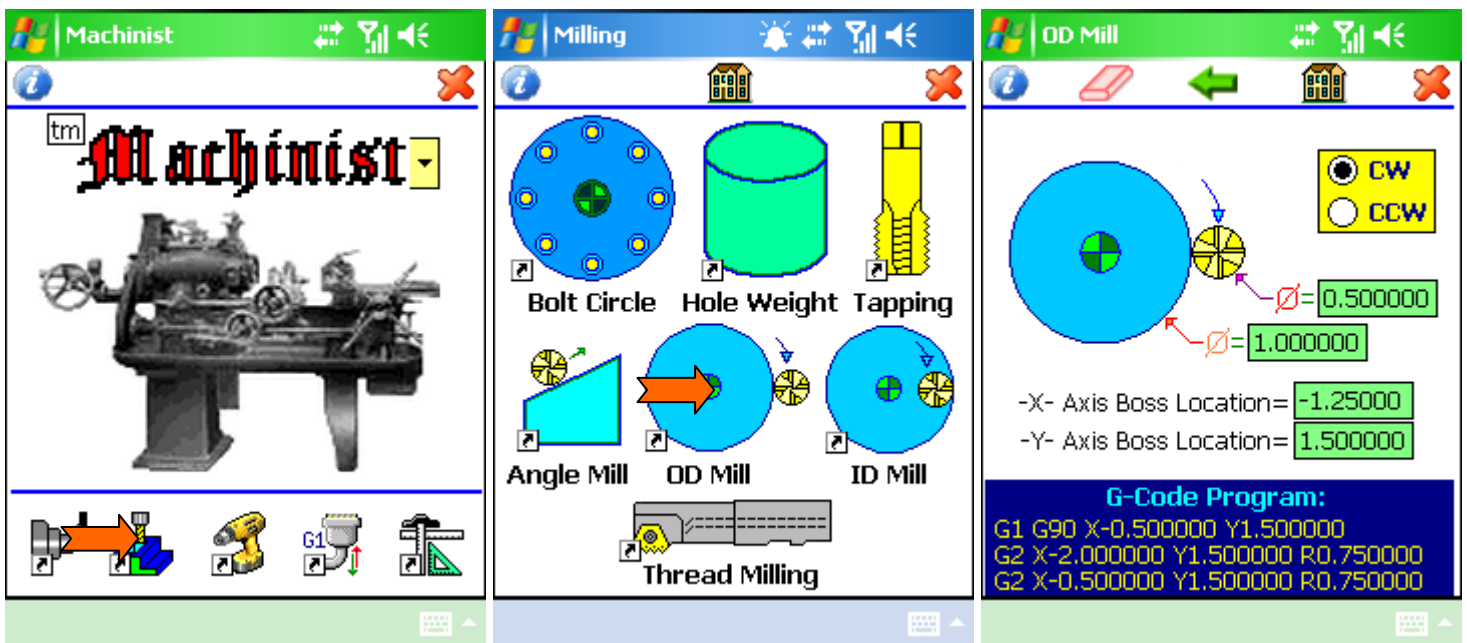
Tap  at any time to access the onboard step by step instructions.

Tap  to clear your information.

Tap  to return to Milling main.

Tap  to return to **Machinist** main.

Tap  to exit **Machinist**.



## ID Mill:

This app generates EIA programming code for pocket milling.

Tap the milling short cut on the main page, or select from the drop down menu.

Tap the ID Mill short cut.


Select CW (clockwise) or CCW (counter clockwise) cutter path.


Enter the diameter of the tool being used.


Enter the bore diameter to be milled.


Enter the -X- axis hole location.


Enter the -Y- axis hole location.

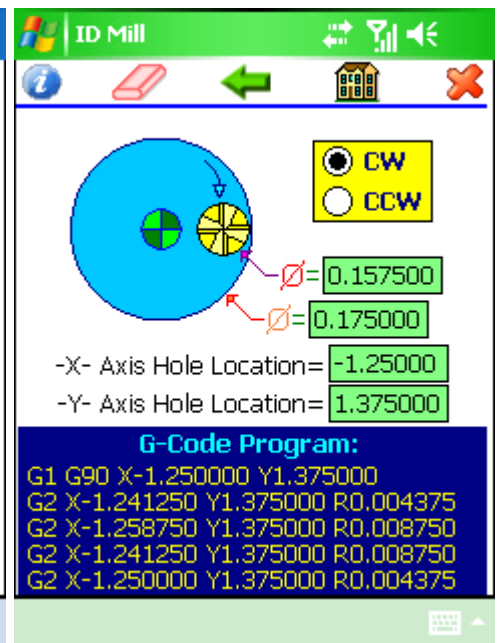
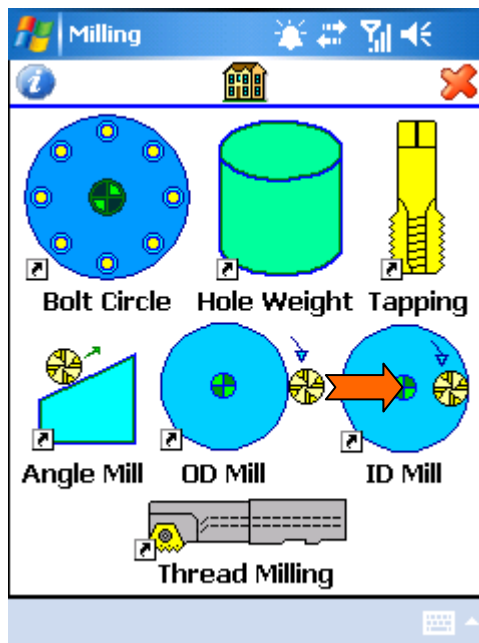
Tap  at any time to access the onboard step by step instructions.

Tap  to clear your information.

Tap  to return to Milling main.

Tap  to return to **Machinist** main.

Tap  to exit **Machinist**.



## Thread Milling:

This app is used to ease EIA thread mill programming.

Tap the milling short cut on the main page, or select from the drop down menu.

Tap the Thread Milling short cut.

Select OD or ID thread milling.

Enter the diameter of the thread.


Enter the length or depth of thread.


Enter the feedrate per insert tooth (i.e. .003).


Enter the diameter of the thread mill.


Enter the recommended SFM (surface feet per minute) for your material type.

Enter The TPI (threads per inch).

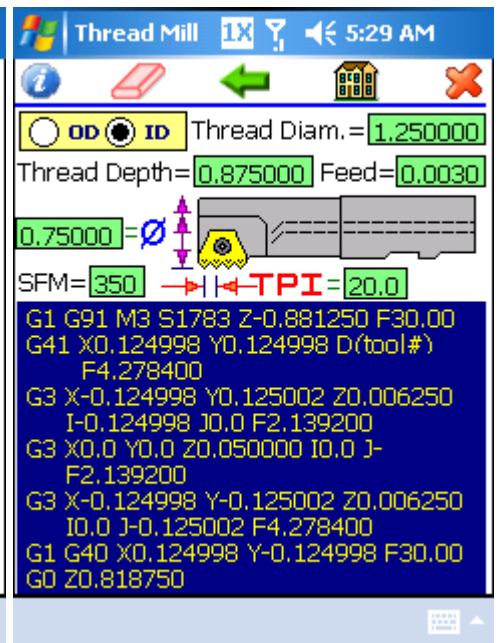
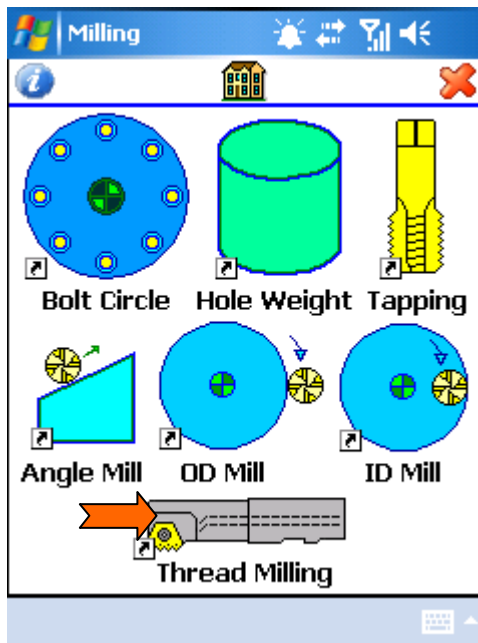
Tap  at any time to access the onboard step by step instructions.

Tap  to clear your information.

Tap  to return to Milling main.

Tap  to return to **Machinist** main.

Tap  to exit **Machinist**.




## Center Drill Tip Comp.:


This app calculates the depth needed to create a specific chamfer size.


Tap the drilling short cut on the main page, or select from the drop down menu.


Tap the Center Drill Tip Comp. short cut.

Enter the required chamfer diameter.

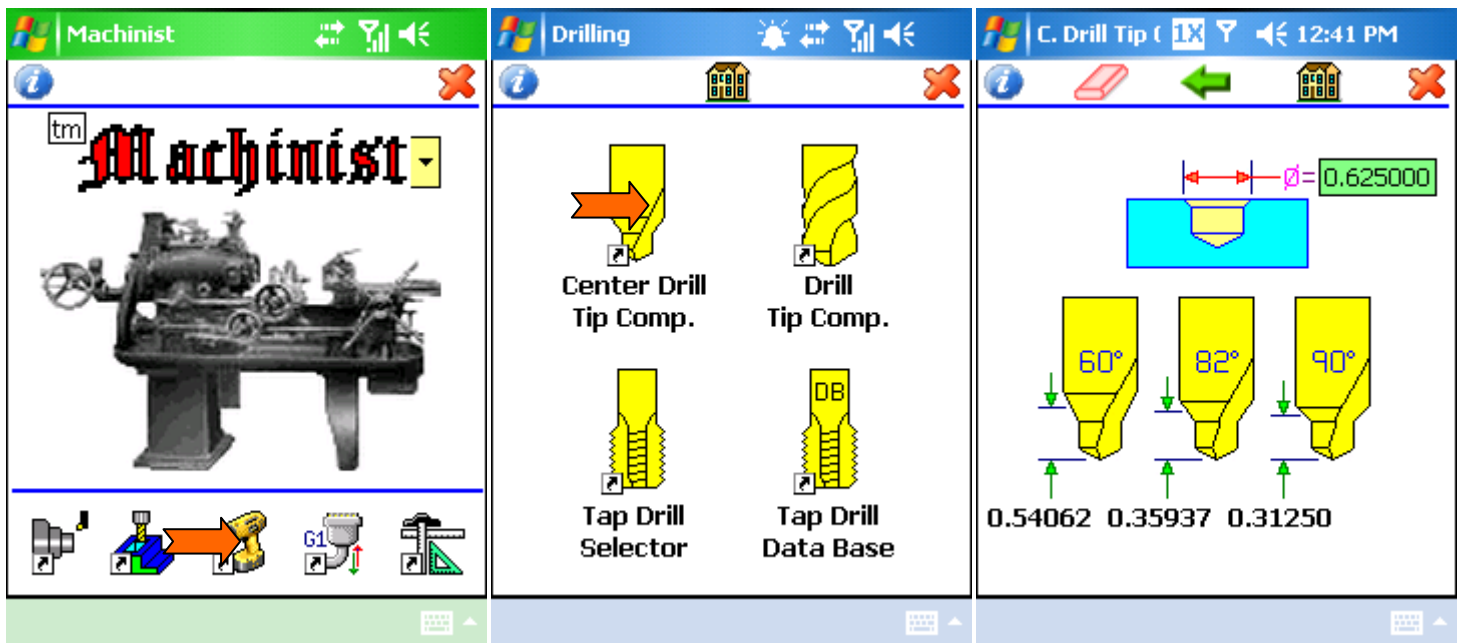
Tap  at any time to access the onboard step by step instructions.

Tap  to clear your information.

Tap  to return to Drilling main.

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.


Tap the drilling short cut on the main page, or select from the drop down menu.


Tap the Drill Tip Comp. short cut.


Enter the drill diameter.

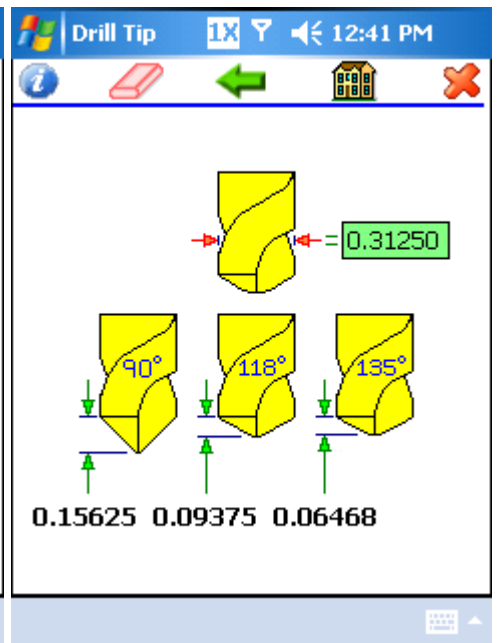
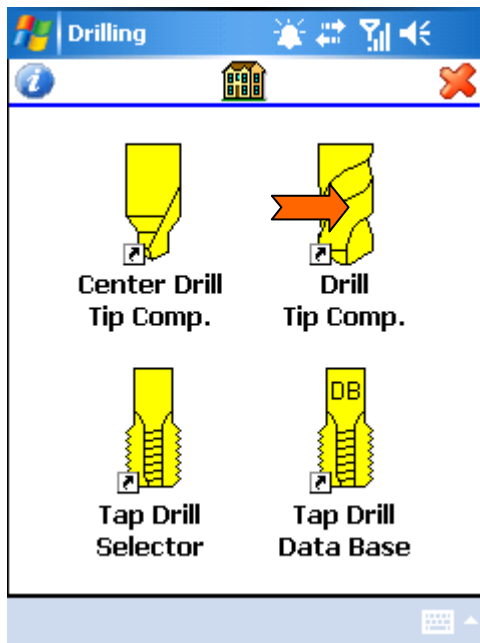
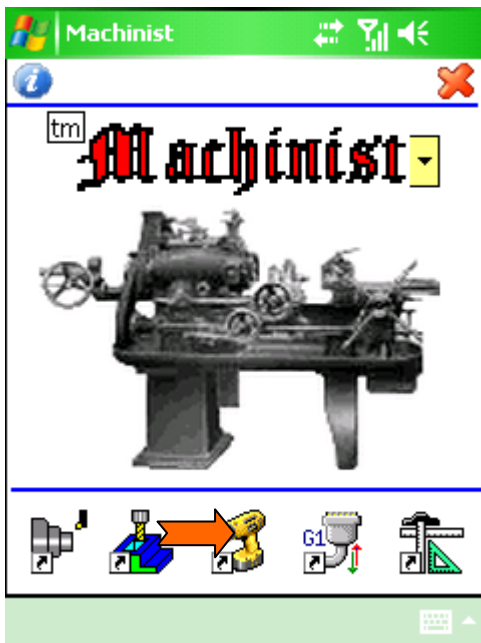
Tap  at any time to access the onboard step by step instructions.

Tap  to clear your information.

Tap  to return to Drilling main.

Tap  to return to **Machinist** main.

Tap  to exit **Machinist**.





## Tap Drill Selector:

### TOP OF THE SCREEN:

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

Tap the drilling short cut on the main page, or select from the drop down menu.

Tap the Tap Drill Selector short cut.

Select the style of tap to use (inch or metric).

Enter the thread diameter.

Enter the thread pitch (threads per inch).

Enter the percent of full threads required (typically 75%).


Read the drill size information.


### BOTTOM OF THE SCREEN:


Select the Tap Type from the drop down menu.


Use the right scroll bar to view all of the tap data.

Tap **Db** to view/edit the Tap Drill Data Base.

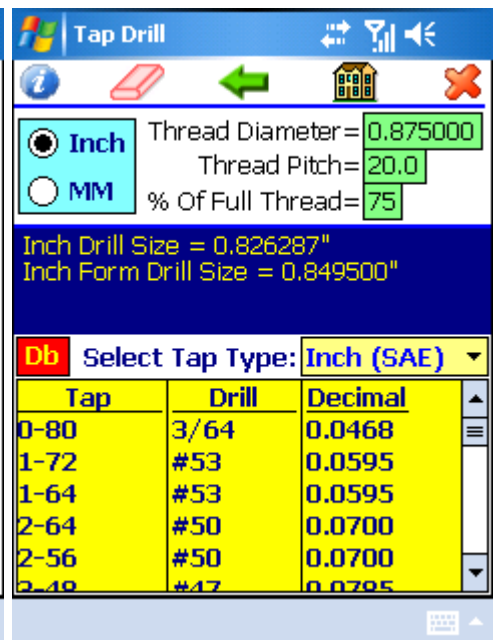
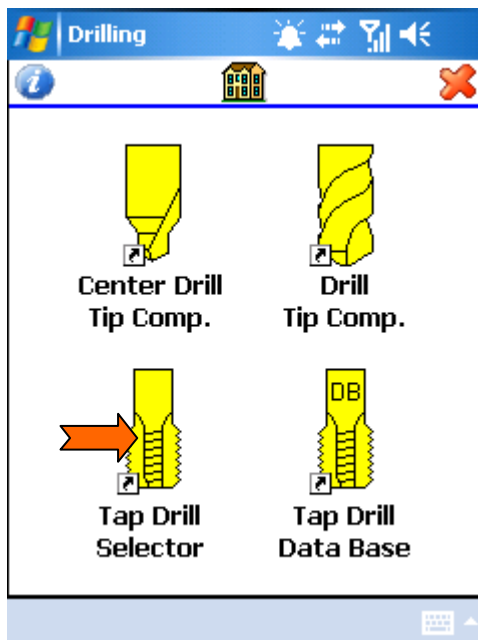
Tap  at any time to access the onboard step by step instructions.

Tap  to clear your information.

Tap  to return to Drilling main.

Tap  to return to **Machinist** main.

Tap  to exit **Machinist**.



## Tap Drill Data Base:

This app is used to enter / edit all of your taps drill sizes into the **Machinist** data base.


Tap the drilling short cut on the main page, or select from the drop down menu.


Tap the Tap Drill Data Base short cut.


Select Edit Enable to add / modify the data base. Select Tap Drill to return to Tap Drill main.


Select the category type for the tap being entered.


Enter the tap size, the tap drill size, and the decimal equivalent of the tap drill.


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

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


Tap  to add a new tap to the data base.


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


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


Tap  to undo a change. Only (1) undo is possible.

Tap  to delete an entry from the data base. **WARNING: This action CAN NOT be undone.**

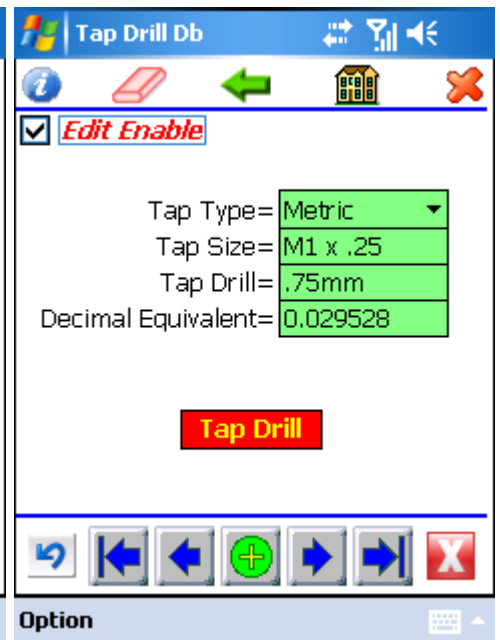
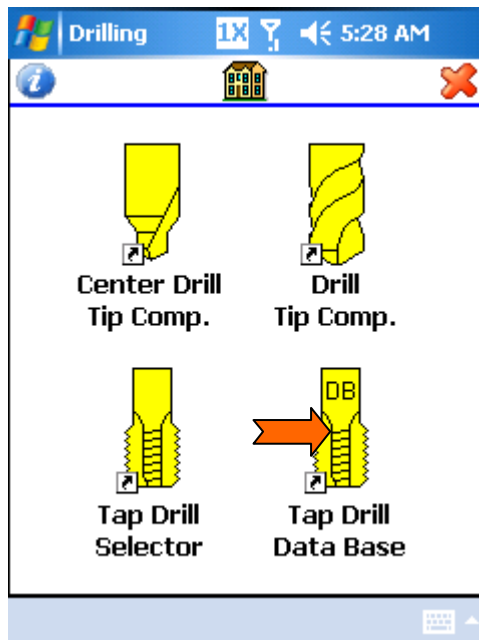
Tap  at any time to access the onboard step by step instructions.

Tap  to clear your information.

Tap  to return to Drilling main.

Tap  to return to **Machinist** main.

Tap  to exit **Machinist**.



## DNC:

This app was designed to transfer a single CNC program from one machine into another.

DNC has a maximum file transfer size limit of 5K.

DNC only works on newer Fanuc style controls, due to the CNC processor speed.

Your CNC must have the comm. port parameters set to 4800 baud rate, 7 bits, even parity, & 2 stop bits. Refer to your machines parameter manual.

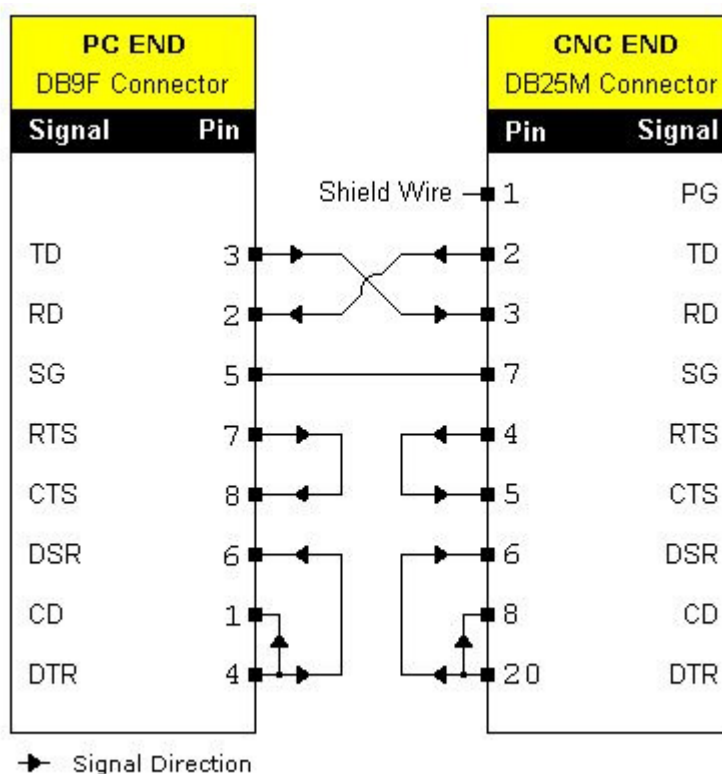
You must purchase or make a communication cable that will interface between the PocketPC and your CNC machine.

**WARNING: Improper wiring may short out the serial port on your device!**

You can purchase a serial cable for most PocketPC's that has a DB9 serial connector.


You will need a DB9 connector kit (the opposite connector of your serial cable), a DB25 male kit (to connect to the CNC), and whatever length wire you want to use (preferably 25ft or less).

This is a typical CNC patch cable wiring schematic:



Tap the DNC short cut on the main page, or select from the drop down menu.

### **Sending to the CNC (uploading):**

Tap  to import a CNC text file (must be named Program.txt) from ActiveSync.

**NOTE:** The first line of your imported program **MUST** be:

ProgramA,ProgramB,ProgramC,ProgramD,ProgramE

The imported program must start and end with quotes (“”).

Get the CNC ready to receive / read a program.

Tap  to upload your program.


Tap yes to proceed or no to abort the upload.

### **Receiving from the CNC (down loading):**


Tap  to receive your program


Tap yes to proceed or no to abort the download.


Press send / punch on the CNC to download your program (Program.txt).

Tap  to save your program on the next Sync (Program.txt).

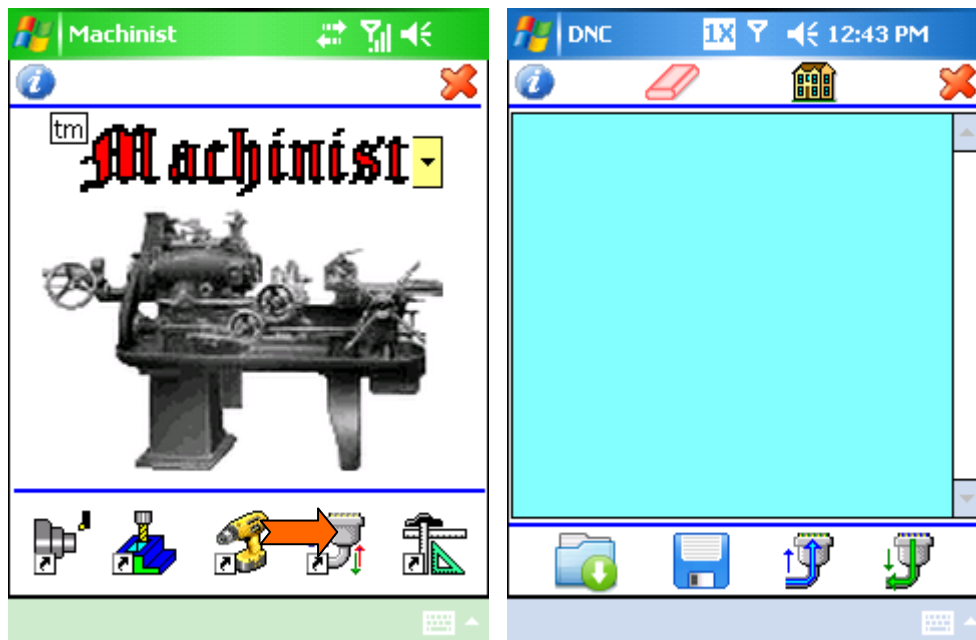
Tap  to access the onboard step by step instructions.

Tap  to clear your information.

Tap  to return to Formulas main.

Tap  to return to **Machinist** main.

Tap  to exit **Machinist**.



## Bar Calc:

This app calculates the part weight and parts per bar of cut material.

Tap the formula short cut on the main page, or select from the drop down menu.

Tap the Bar Calc button.

Enter the bar length (in feet).

Enter the width of the kerf / cut-off (in inches).


Enter the length of the bar remnant (in inches).


Enter the bar / part diameter.


Enter the cut part length.

Select the material being used.

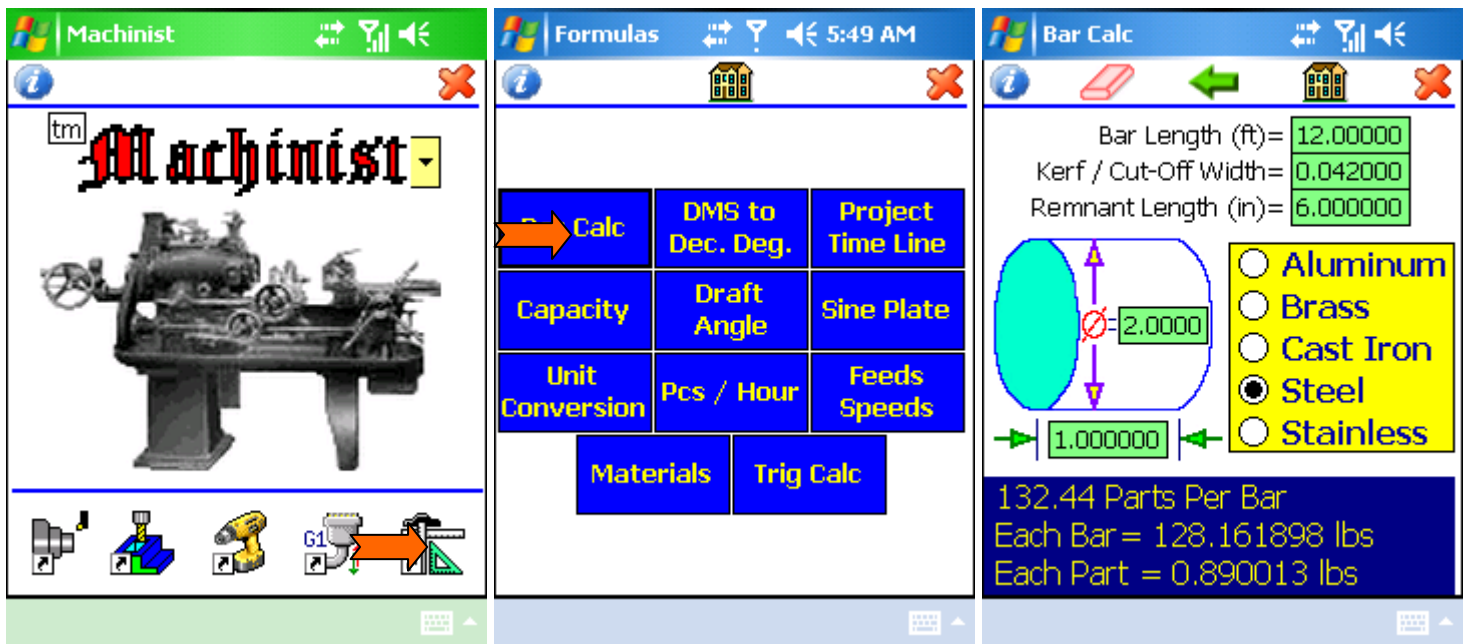
Tap  to access the onboard step by step instructions.

Tap  to clear your information.

Tap  to return to Formulas main.

Tap  to return to **Machinist** main.

Tap  to exit **Machinist**.




## DMS to Dec. Deg.


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


Tap the formula short cut on the main page, or select from the drop down menu.


Select 'deg,min,sec' or Decimal Degree input.

Enter your information in deg,min,sec or decimal degrees.

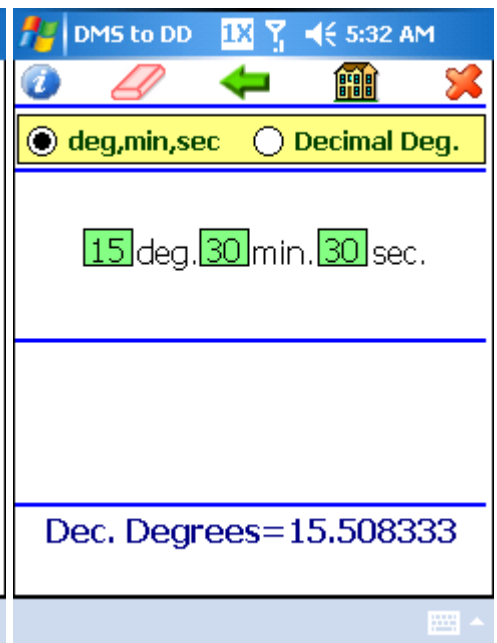
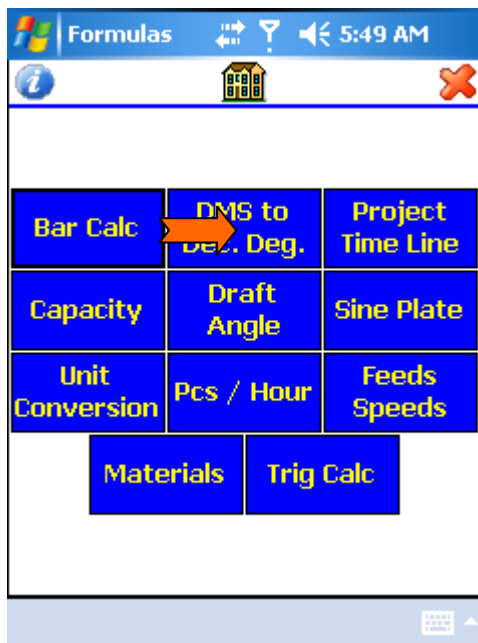
Tap  at any time to access the onboard step by step instructions.

Tap  to clear your information.

Tap  to return to Formulas main.

Tap  to return to **Machinist** main.

Tap  to exit **Machinist**.



## Project 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 the formula short cut on the main page, or select from the drop down menu.


Select ‘# of weeks’ or ‘Ending Date’ input method.

Select the start date of your project.

Enter the length of the project (in weeks), or the ending date of the project.

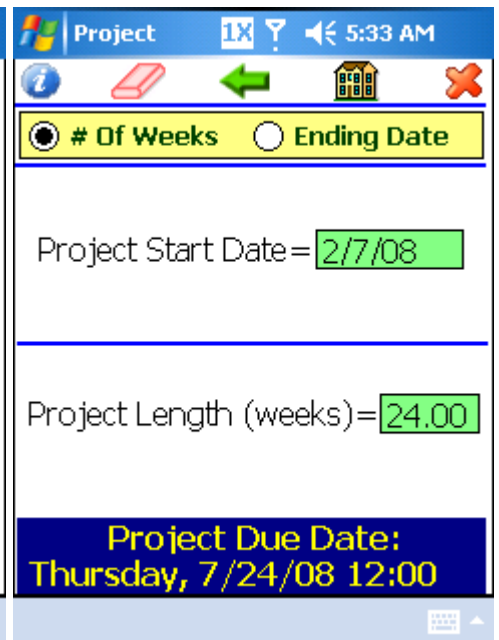
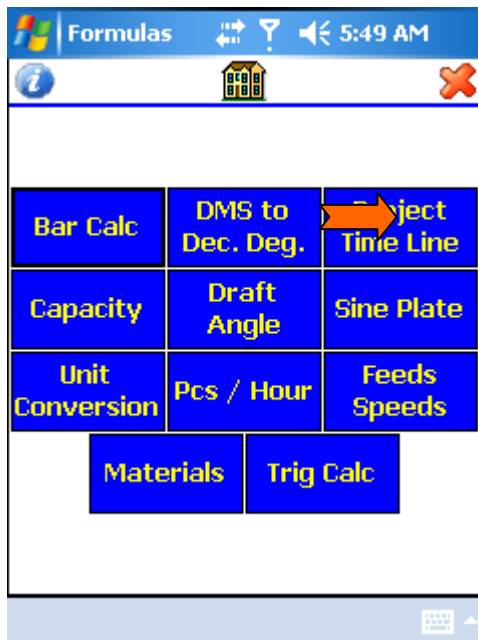
Tap  at any time to access the onboard step by step instructions.

Tap  to clear your information.

Tap  to return to Formulas main.

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.

Tap the formula short cut on the main page, or select from the drop down menu.

Tap the Capacity button.

Enter the customers annual parts requirement.

Enter the number of work weeks in a year.


Enter the quoted pieces per hour.


Enter the number of days per week worked.


Enter the number of work hours in a day.


Enter the number of machines being utilized.

Select the required efficiency (100% = full capacity).

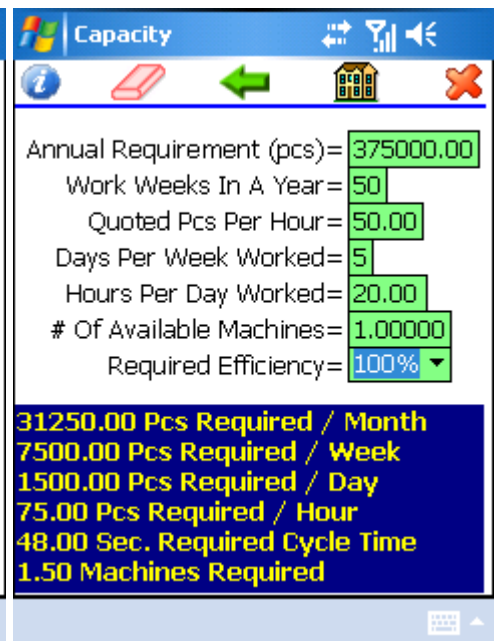
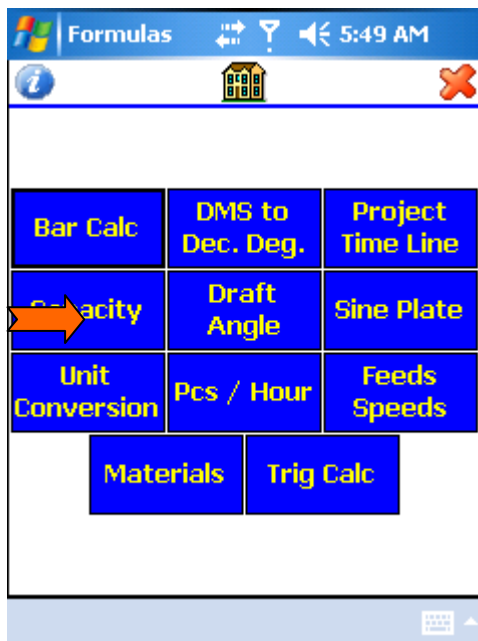
Tap  at any time to access the onboard step by step instructions.

Tap  to clear your information.

Tap  to return to Formulas main.

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.


Tap the formula short cut on the main page, or select from the drop down menu.


Tap the Draft Angle button.


Enter the draft angle in decimal degrees.


Enter the length of draft angle.

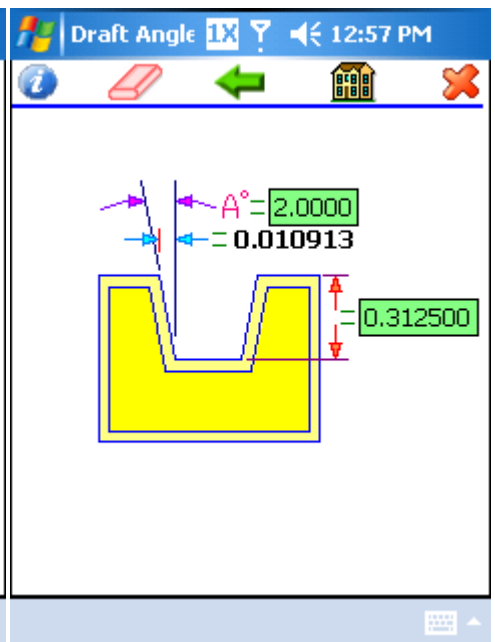
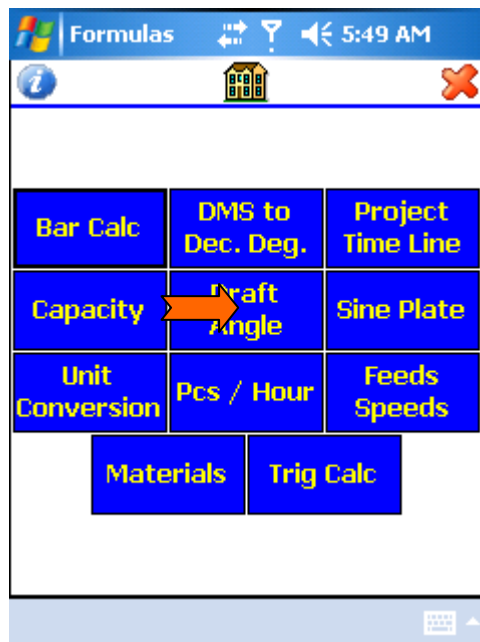
Tap  at any time to access the onboard step by step instructions.

Tap  to clear your information.

Tap  to return to Formulas main.

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.


Tap the formula short cut on the main page, or select from the drop down menu.


Tap the Sine Plate button.


Enter the required angle in decimal degrees.


Enter the sine plate base length.

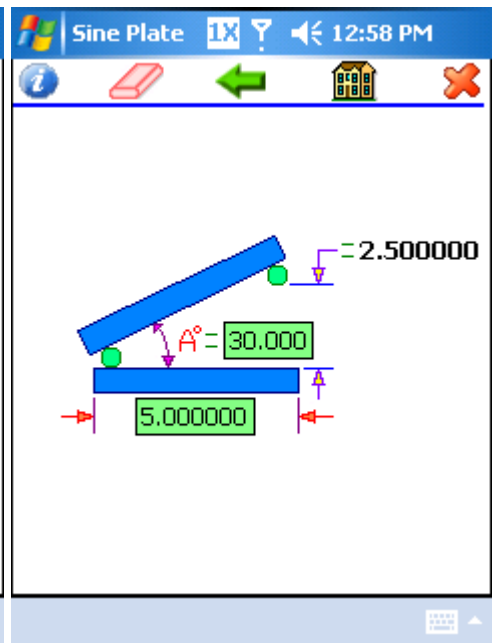
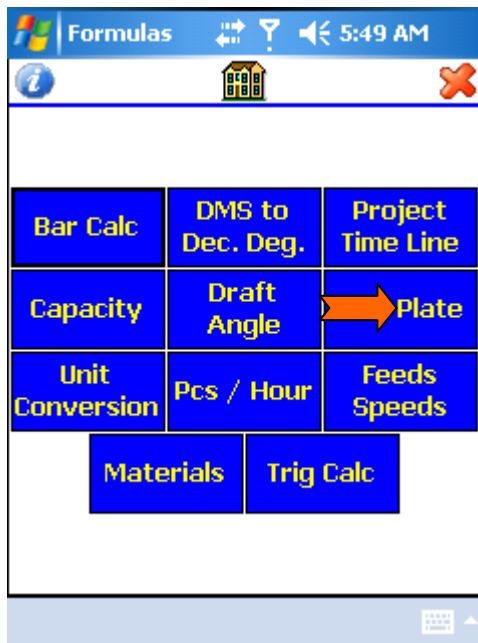
Tap  at any time to access the onboard step by step instructions.

Tap  to clear your information.

Tap  to return to Formulas main.

Tap  to return to **Machinist** main.

Tap  to exit **Machinist**.



## Unit Conversion:

This app converts a number from one category into another.


**TIP:** Conversion is helpful on machinery that has metric pressure gages.


Tap the formula short cut on the main page, or select from the drop down menu.


Tap the Unit Conversion button.

Enter the value to be converted.

Select the preferred conversion.

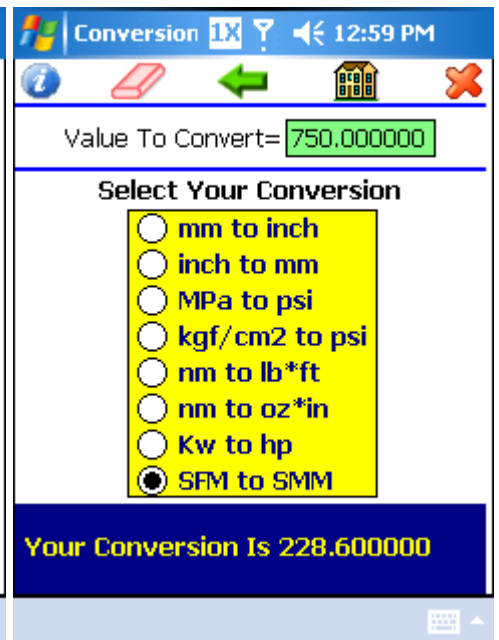
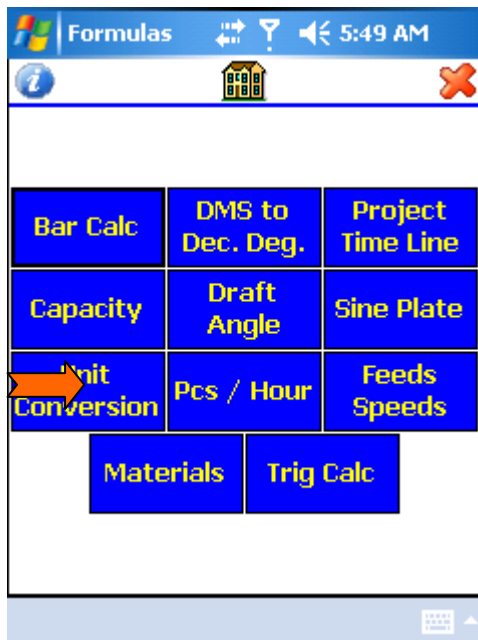
Tap  at any time to access the onboard step by step instructions.

Tap  to clear your information.

Tap  to return to Formulas main.

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 the formula short cut on the main page, or select from the drop down menu.

Tap the Pcs / Hour button.

Select whether the time input method will be from the stopwatch or manual input.

Tap the green stopwatch button and tap, then ok to start the stopwatch.

Tap the red stopwatch button, then ok to stop the stopwatch.

The total cycle time from the stopwatch (in seconds) will automatically be used for all calculations if stopwatch is selected as the input method.

If manual input is selected, enter your cycle time where provided.

Enter the number of parts per load.


Enter the number of machines being utilized.


Enter the total number of hours per day worked.


Enter the number of work days in a week.


Enter the number of weeks worked in a year.

Select the percentage of required efficiency (100% = full capacity).

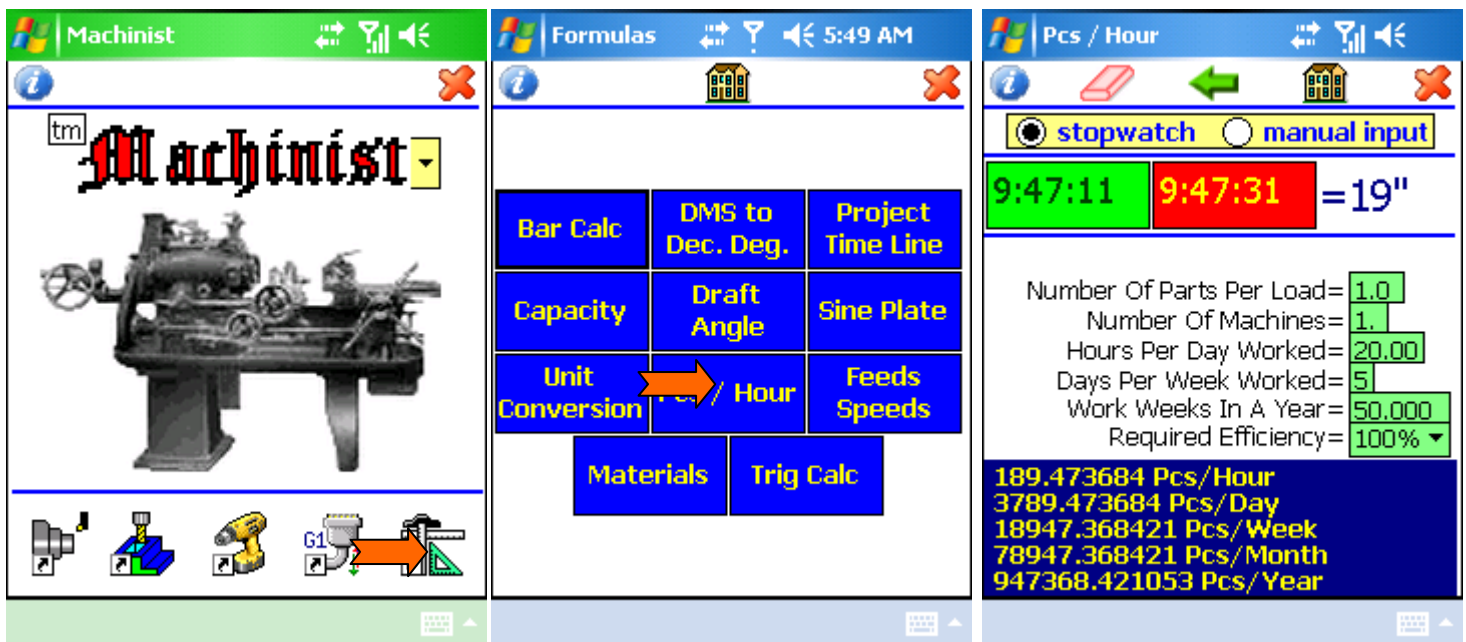
Tap  at any time to access the onboard step by step instructions.

Tap  to clear your information.

Tap  to return to Formulas main.

Tap  to return to **Machinist** main.

Tap  to exit **Machinist**.



## Feeds-Speeds:

This app calculates machining feeds and speeds.

Tap the formula short cut on the main page, or select from the drop down menu.

Tap the Feeds-Speeds button.


To calculate SFM (surface feet per minute): Enter the RPM (revolutions per minute) & the tool diameter.


To calculate RPM (revolutions per minute): Enter the SFM (surface feet per minute) & the tool diameter.


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


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

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

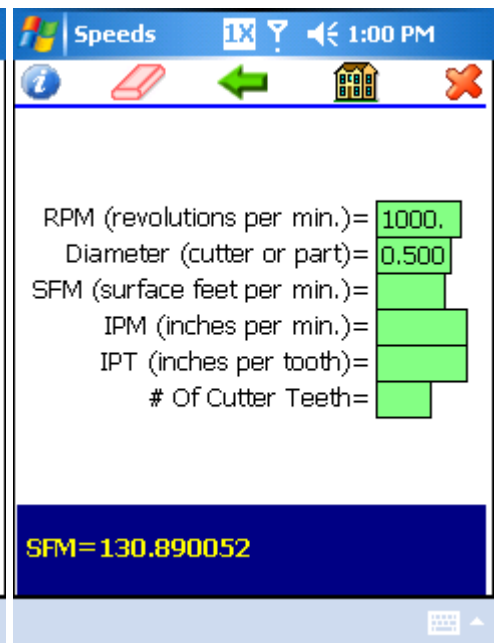
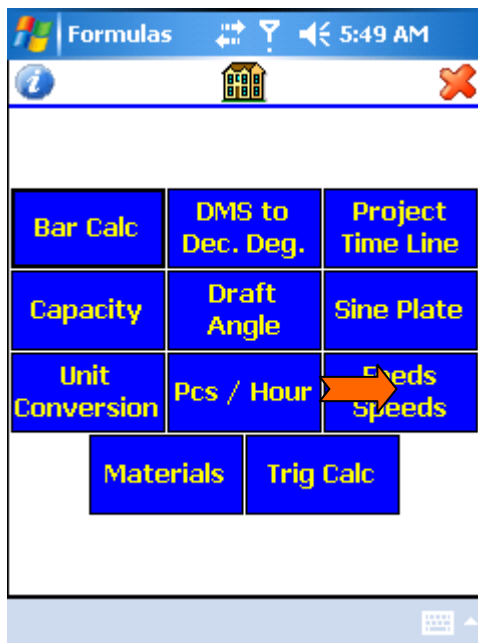
Tap  at any time to access the onboard step by step instructions.

Tap  to clear your information.

Tap  to return to Formulas main.

Tap  to return to **Machinist** main.

Tap  to exit **Machinist**.



## Materials:


This app displays known material cross references by material type.


Tap the formula short cut on the main page, or select from the drop down menu.


Tap the Materials short cut.

Select your material type from the drop down menu.

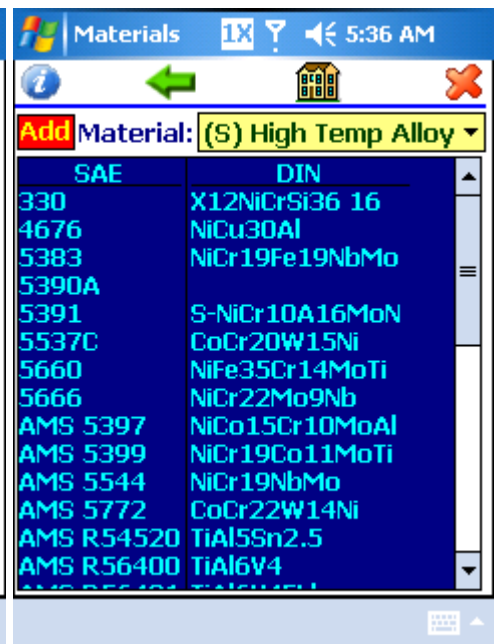
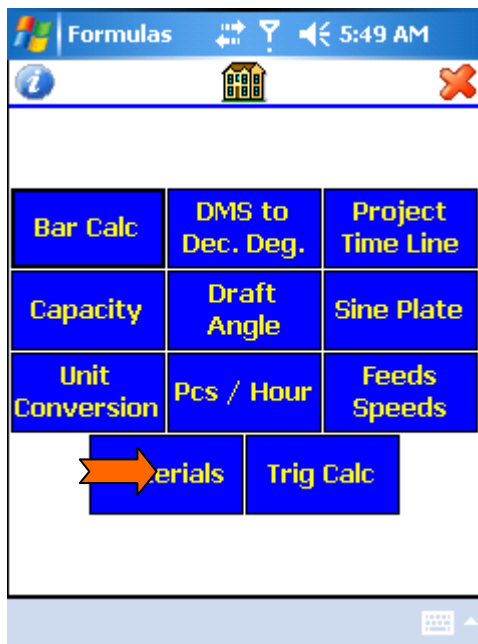
Tap 'Add' to add / edit the Materials data base.

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

Tap  to return to Formulas main.

Tap  to return to **Machinist** main.

Tap  to exit **Machinist**.



## Materials DB:


This app allows you to enter your material specification.

Tap the formula short cut on the main page, or select from the drop down menu.


Tap the Materials short cut, then tap the 'Add' button.

Select 'Edit' to add/ modify the information.


Enter your material information.


Tap  to view/edit the first material in the data base.

Tap  to view/edit the previous material in the data base.


Tap  to add a new material to the data base.


Tap  to view/edit the next material in the data base.


Tap  to view/edit the last material in the data base.


Tap  to undo a change. Only (1) undo is possible.

Tap  to delete an entry from the data base. **WARNING: This action CAN NOT be undone.**

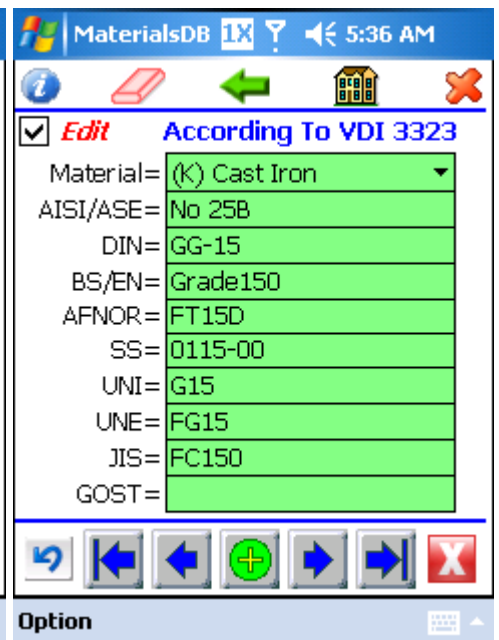
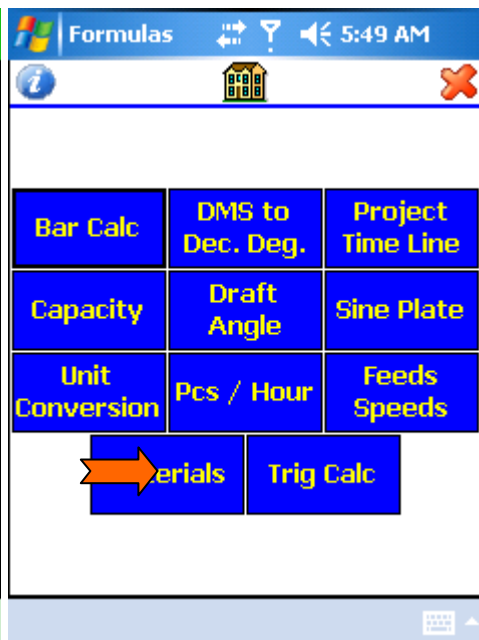
Tap  at any time to access the onboard step by step instructions.

Tap  to clear your information.

Tap  to return to Materials main.

Tap  to return to **Machinist** main.

Tap  to exit **Machinist**.




## Trig Calc:


This app calculates the unknown sides and angles of a right triangle.


Tap the formula short cut on the main page, or select from the drop down menu.


Tap the Trig Calc short cut.

Enter your known information (minimum of one angle and one side).

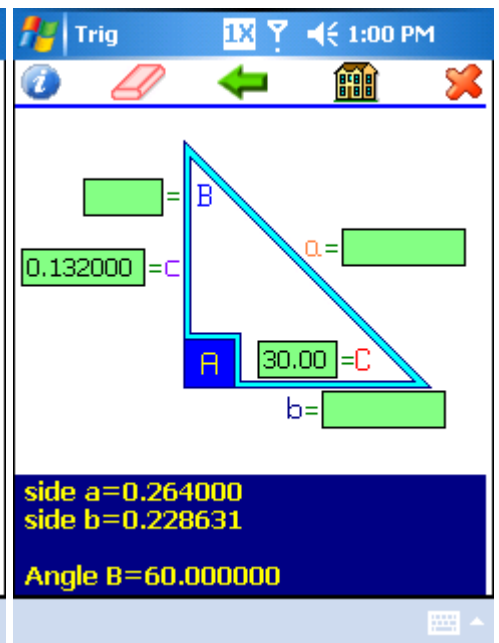
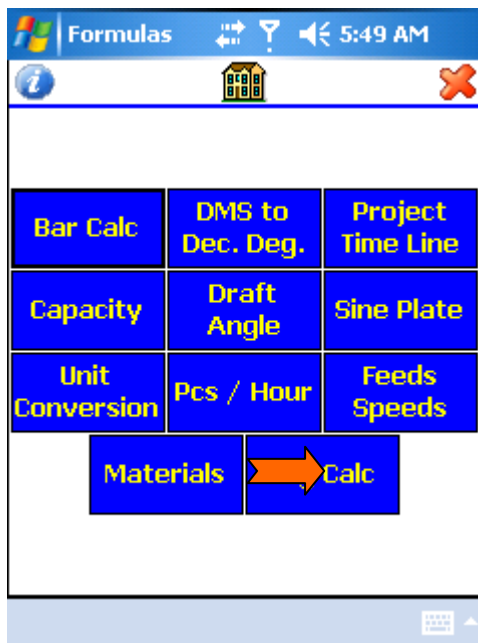
Tap  at any time to access the onboard step by step instructions.

Tap  to clear your information.

Tap  to return to Formulas main.

Tap  to return to **Machinist** main.

Tap  to exit **Machinist**.





## **EULA (End User License Agreement):**

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

You may **not** reproduce or re-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 or guarantees.

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**.