

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.

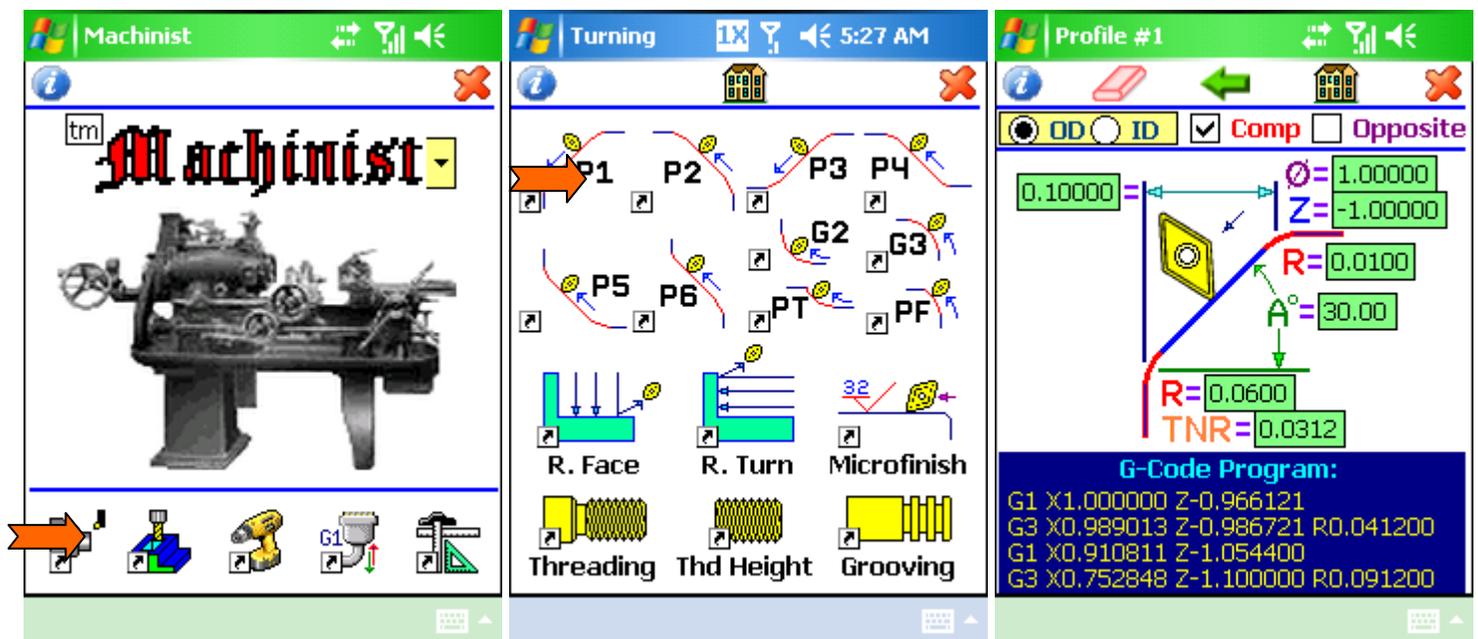
Tap  to clear your information.

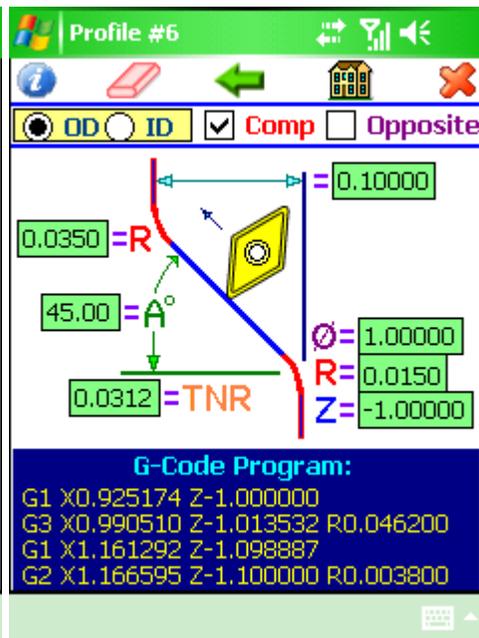
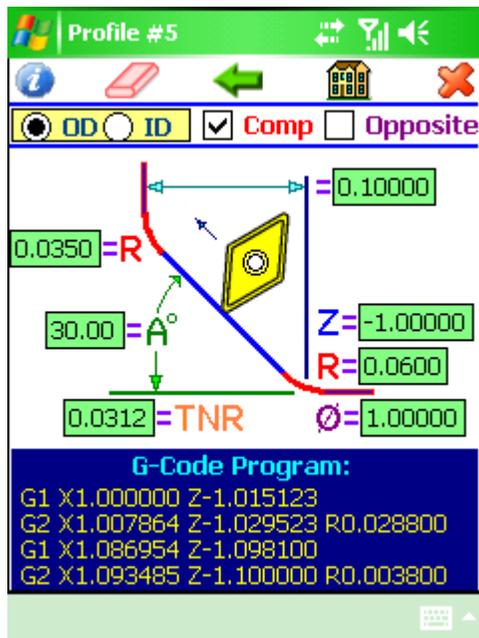
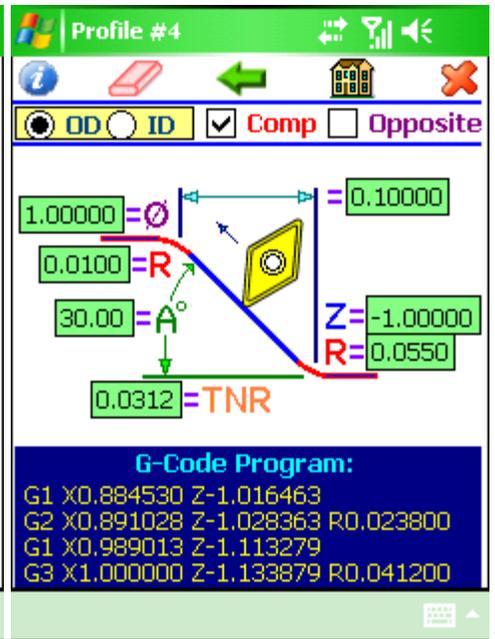
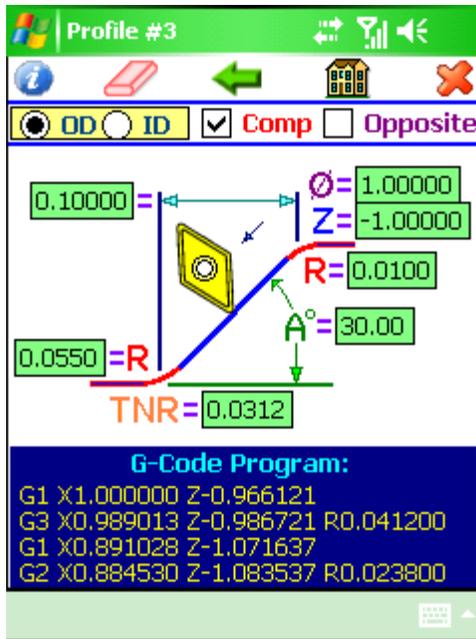
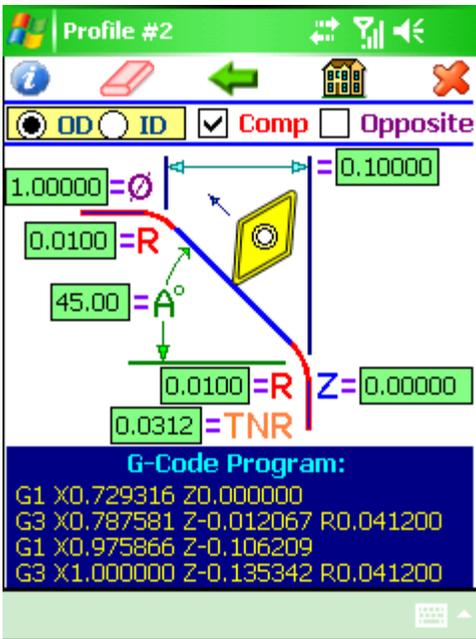
Tap  to return to Turning main.

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.



G2 Radius 1X 8:58 AM

OD ID Comp Opposite

$Z = -1.12500$
 $TNR = 0.0312$
 $R = 0.0400$
 $\varnothing = 1.25000$

G-Code Program:
 G1 X1.250000
 Z-1.116200
 G2 X1.267600 Z-1.125000 R0.008800

G3 Radius 1X 8:59 AM

OD ID Comp Opposite

$\varnothing = 1.25000$
 $R = 0.0400$
 $TNR = 0.0312$
 $Z = -1.12500$

G-Code Program:
 G1 X1.107600
 Z-1.125000
 G3 X1.250000 Z-1.196200 R0.071200

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.

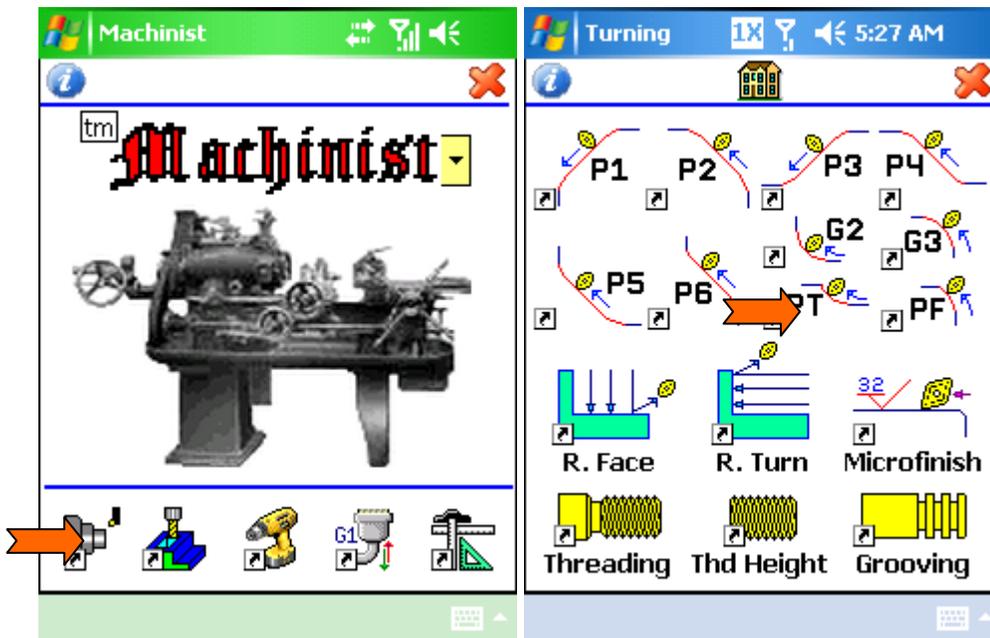
Tap  to clear your information.

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
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.

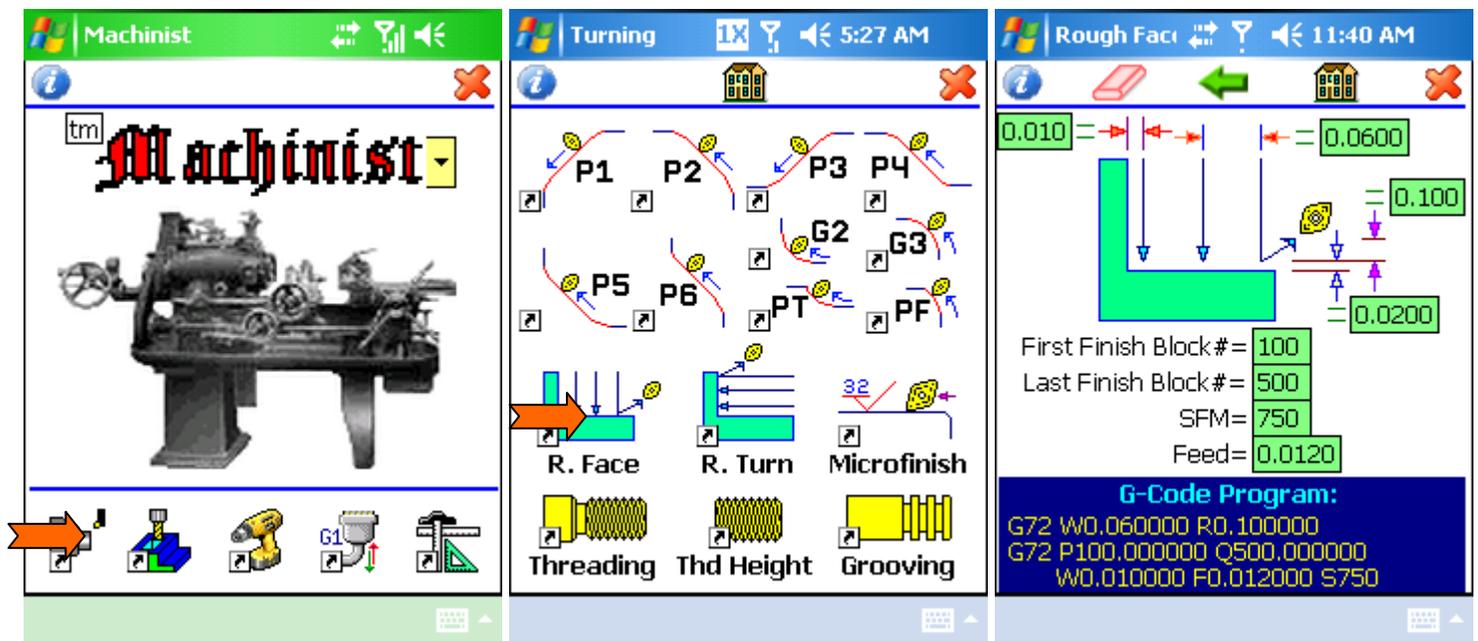
Tap  at any time to access the onboard step by step instructions.

Tap  to clear your information.

Tap  to return to Turning main.

Tap  to return to **Machinist** main.

Tap  to exit **Machinist**.



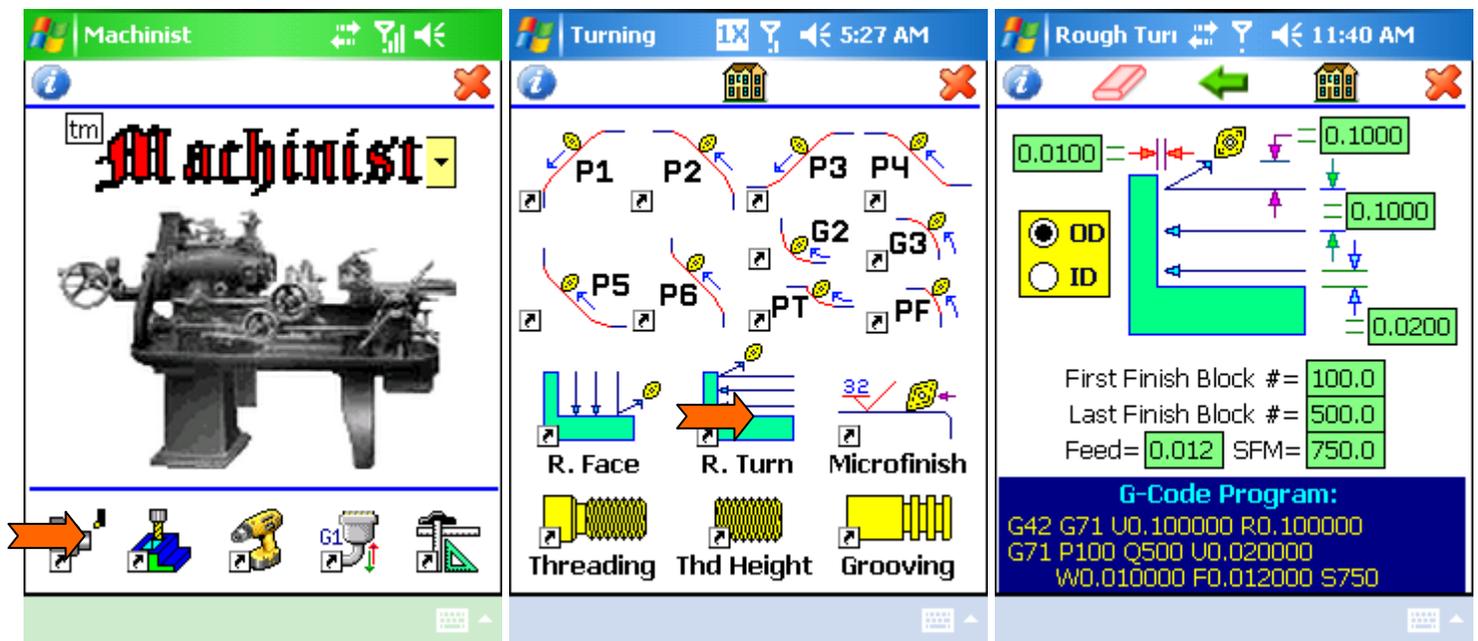
The image displays three screenshots of the Machinist app interface:

- Left Screenshot (Machinist Main):** Shows the main menu with the title "Machinist" and a large image of a lathe. Below the image are several icons representing different machining operations.
- Middle Screenshot (Turning):** Shows the "Turning" screen with a diagram of a lathe part and various operation icons: R. Face, R. Turn, Microfinish, Threading, Thd Height, and Grooving. The time is 5:27 AM.
- Right Screenshot (Rough Face):** Shows the "Rough Face" screen with a diagram of a roughed part and input fields for parameters: First Finish Block # = 100, Last Finish Block # = 500, SFM = 750, and Feed = 0.0120. The time is 11:40 AM. Below the fields is a "G-Code Program" section with the following code:

```
G72 W0.060000 R0.100000
G72 P100.000000 Q500.000000
W0.010000 F0.012000 S750
```

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).

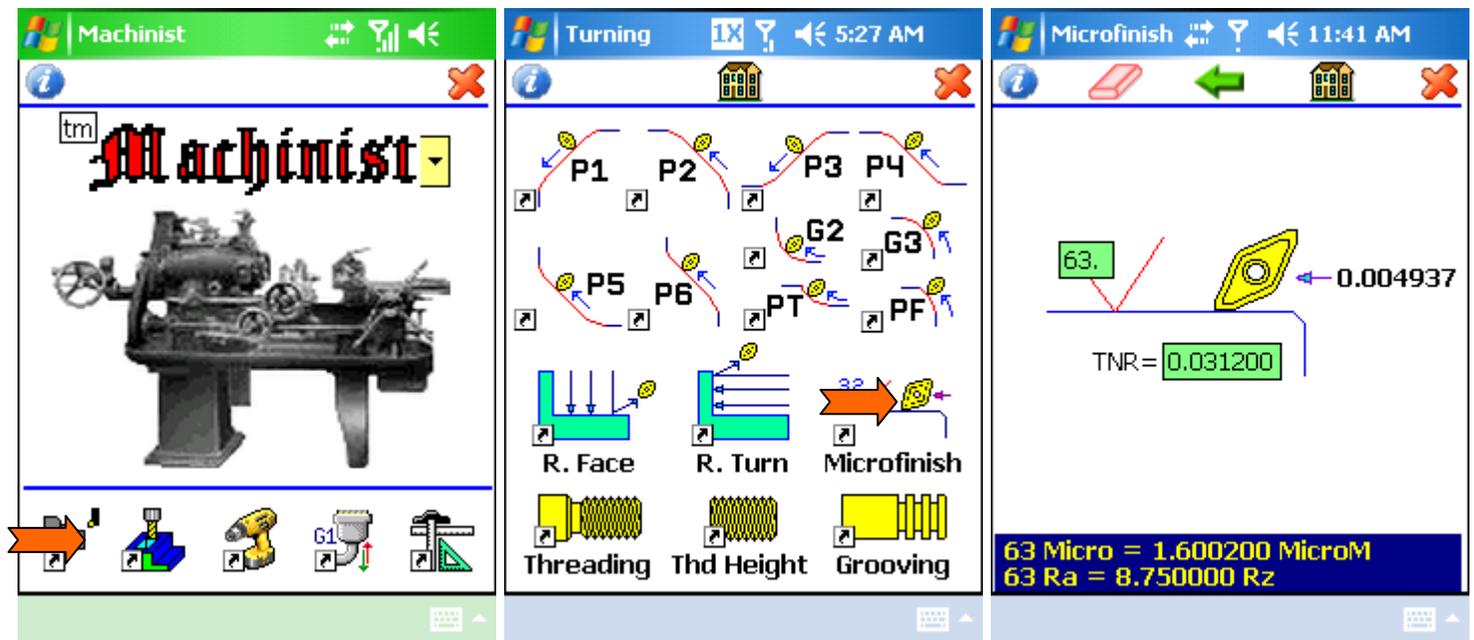
Tap  at any time to access the onboard step by step instructions.

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.

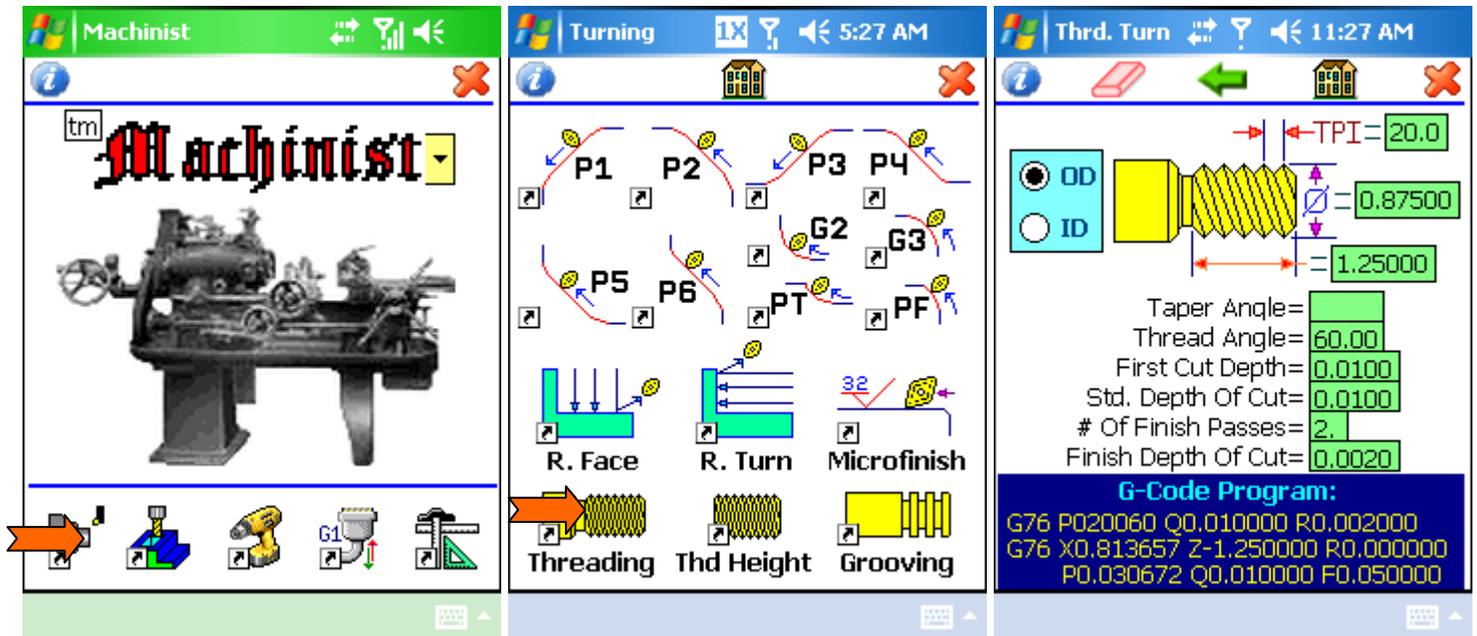
Tap  at any time to access the onboard step by step instructions.

Tap  to clear your information.

Tap  to return to Turning main.

Tap  to return to **Machinist** main.

Tap  to exit **Machinist**.



The image displays three sequential screenshots of the Machinist app's Threading interface:

- Left Screenshot (Machinist Main):** Shows the main menu with a large 'Machinist' logo and a lathe image. A bottom toolbar contains icons for various machining operations, with the Threading icon highlighted by an orange arrow.
- Middle Screenshot (Turning Main):** Shows the 'Turning' main screen with a navigation menu. The Threading icon is highlighted with an orange arrow.
- Right Screenshot (Thrd. Turn Setup):** Shows the Threading setup screen. It features a 3D model of a threaded part with various parameters:
 - OD (Outer Diameter) selected
 - TPI = 20.0
 - Thread Diameter = 0.87500
 - Thread Length = 1.25000
 - Taper Angle = [blank]
 - Thread Angle = 60.00
 - First Cut Depth = 0.0100
 - Std. Depth Of Cut = 0.0100
 - # Of Finish Passes = 2
 - Finish Depth Of Cut = 0.0020
 Below the model, a 'G-Code Program' is displayed:


```
G76 P020060 Q0.010000 R0.002000
G76 X0.813657 Z-1.250000 R0.000000
P0.030672 Q0.010000 F0.050000
```

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).

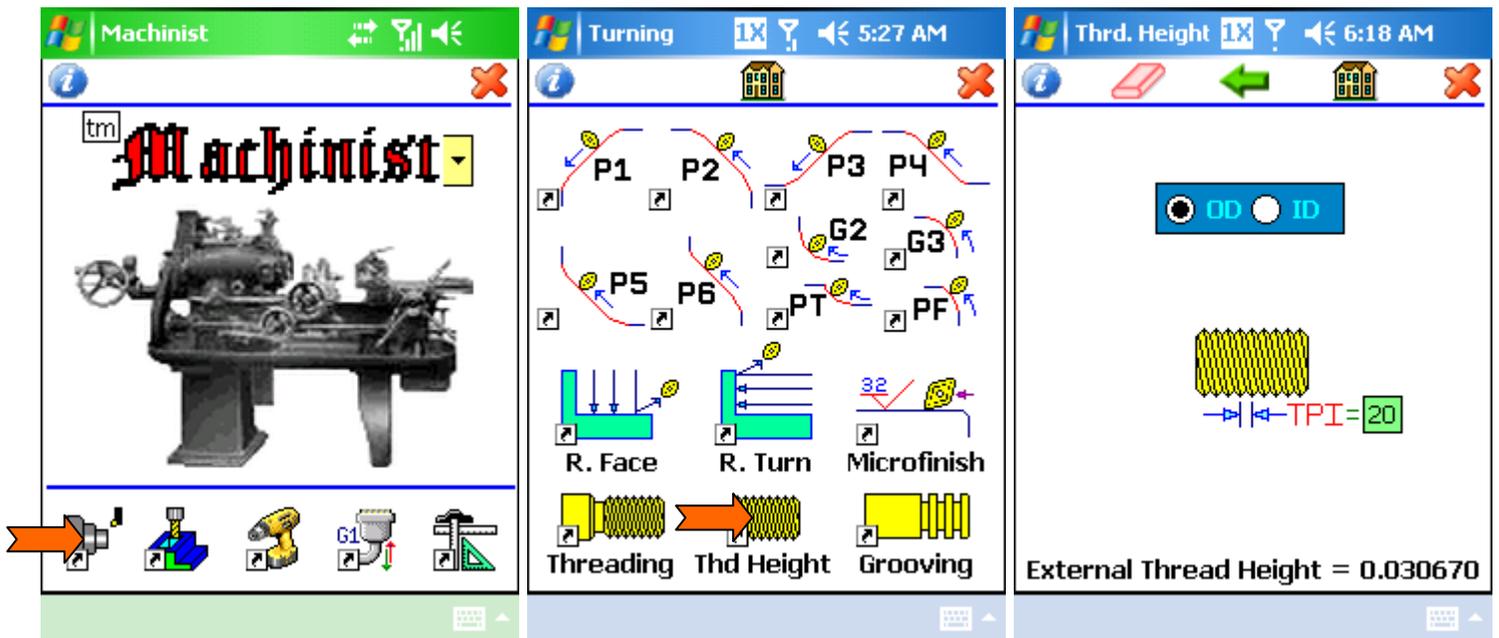
Tap  at any time to access the onboard step by step instructions.

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.

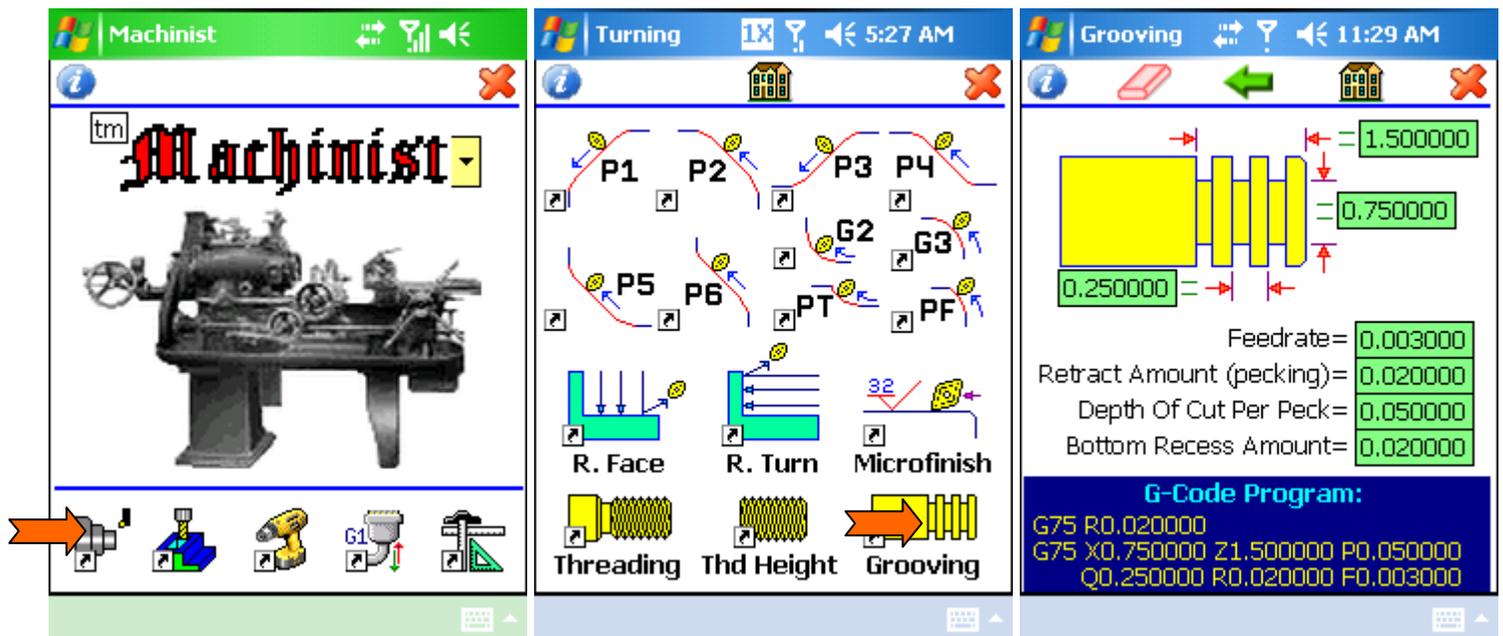
Tap  at any time to access the onboard step by step instructions.

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.

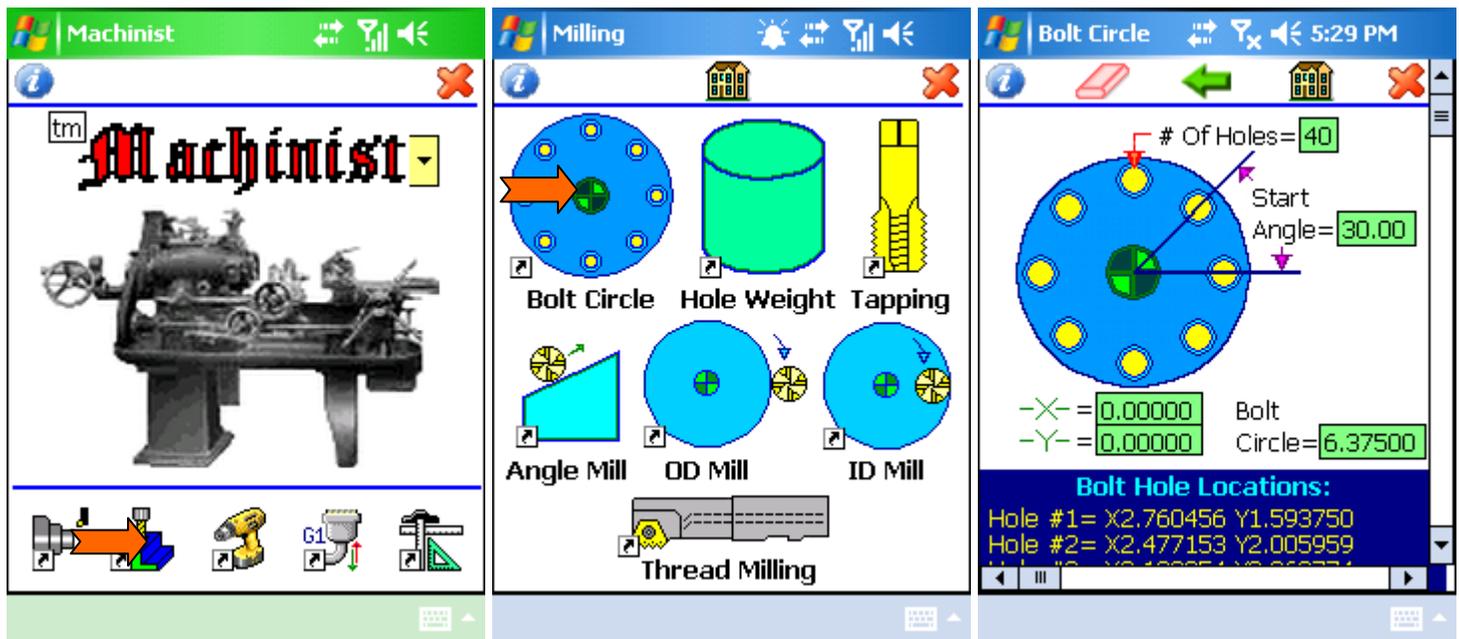
Tap  at any time to access the onboard step by step instructions.

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.

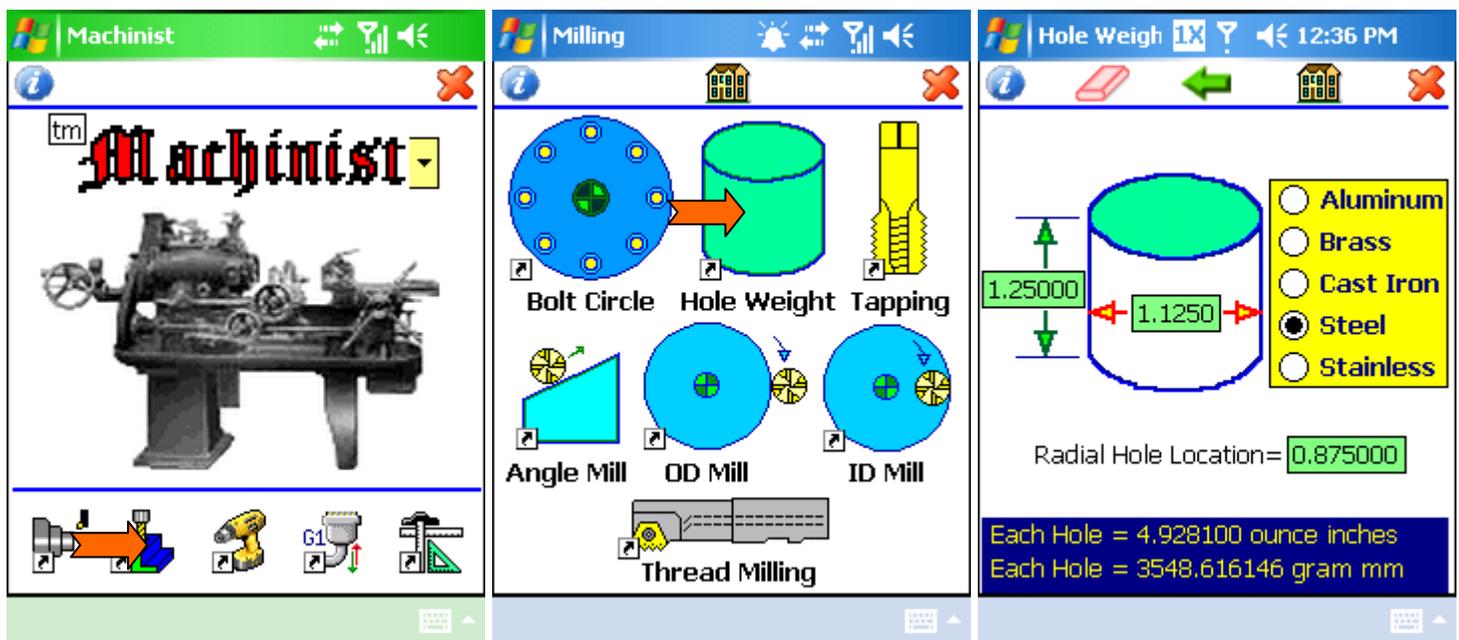
Tap  at any time to access the onboard step by step instructions.

Tap  to clear your information.

Tap  to return to Milling main.

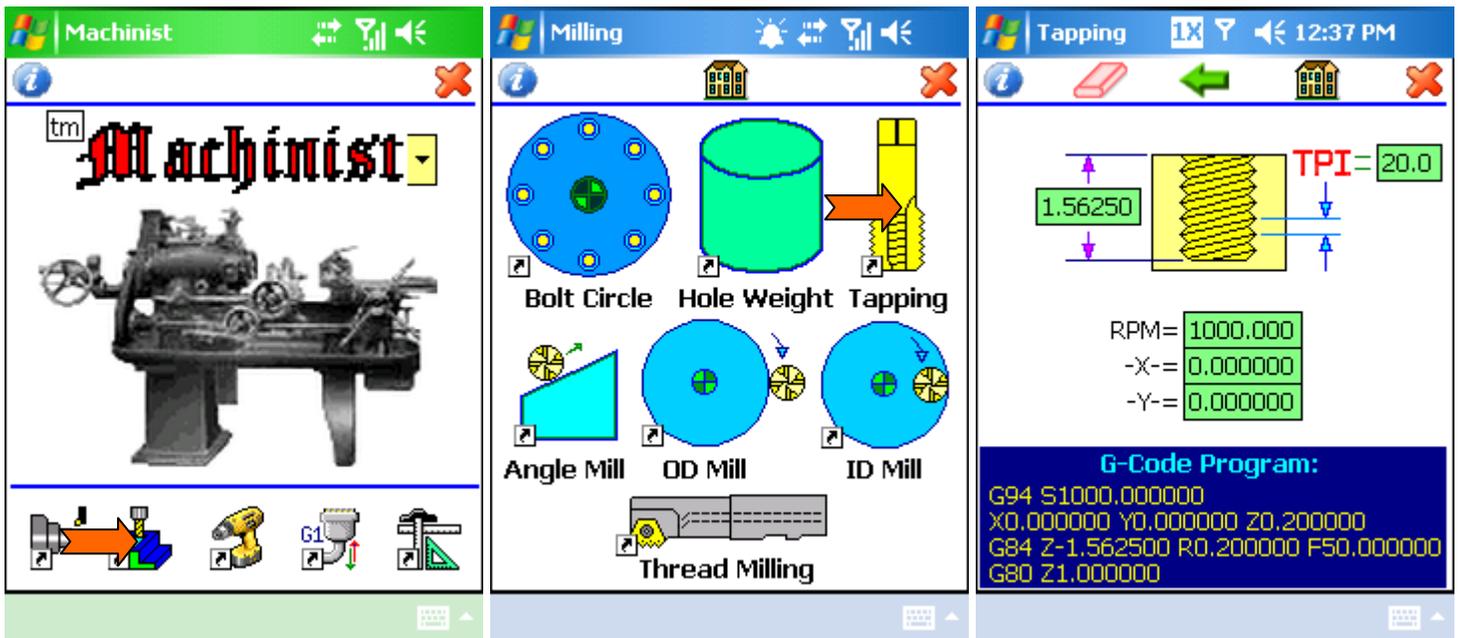
Tap  to return to **Machinist** main.

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.

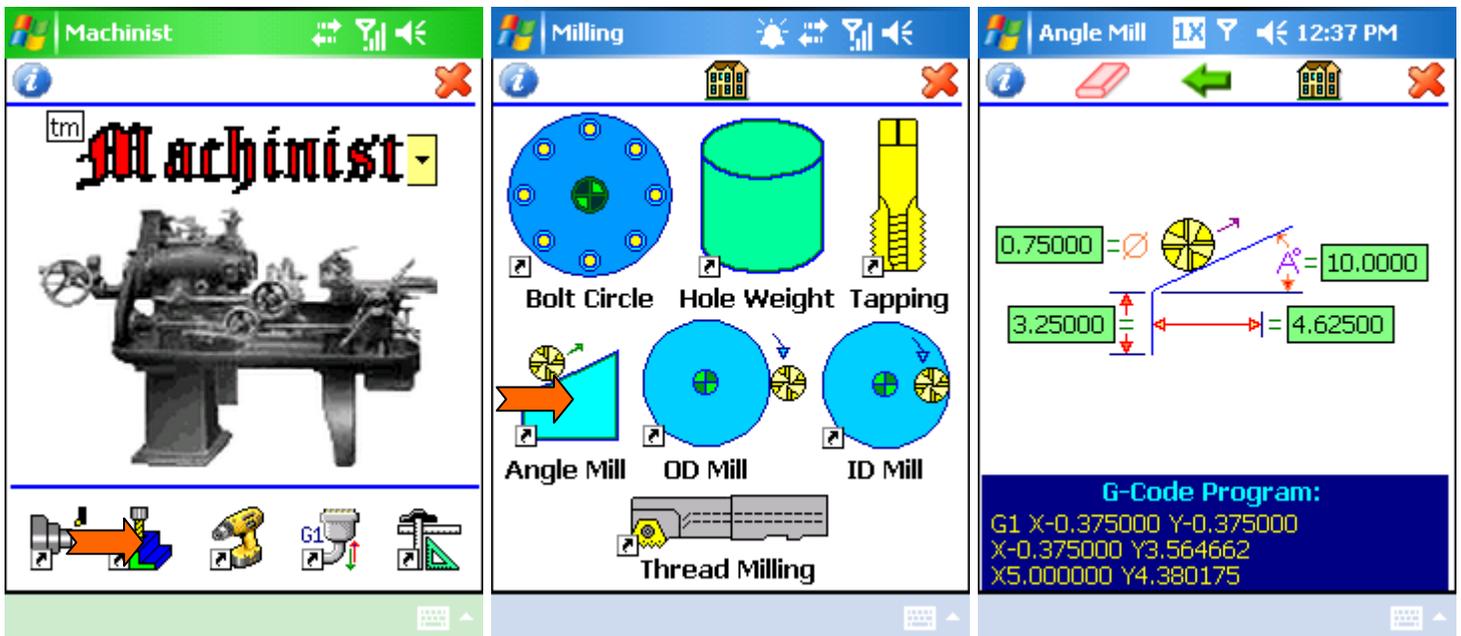
Tap  at any time to access the onboard step by step instructions.

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.

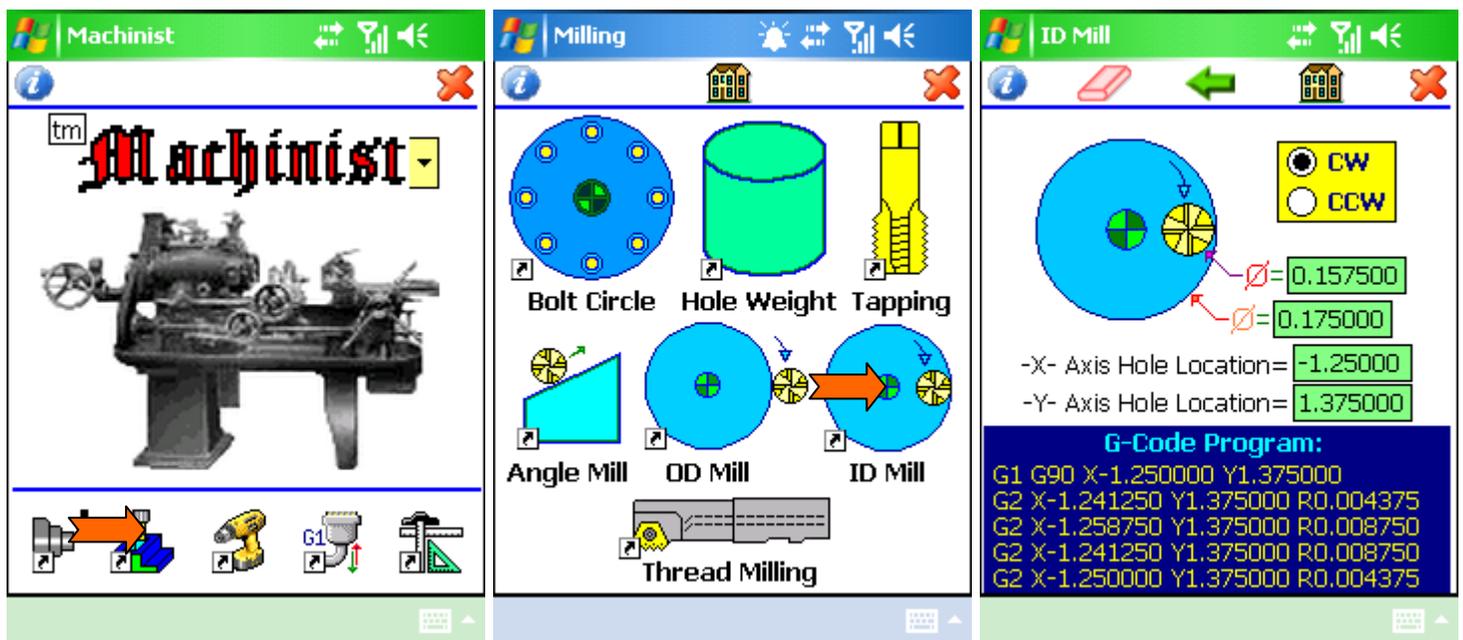
Tap  at any time to access the onboard step by step instructions.

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).

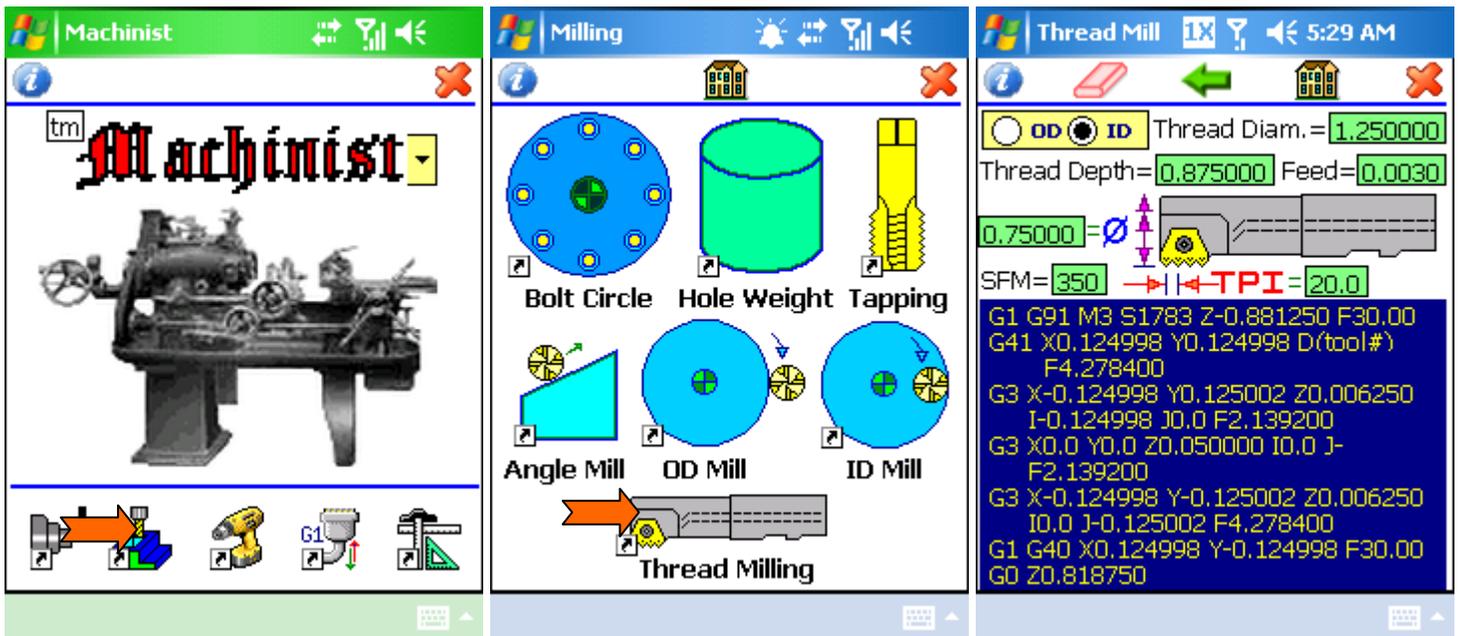
Tap  at any time to access the onboard step by step instructions.

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.

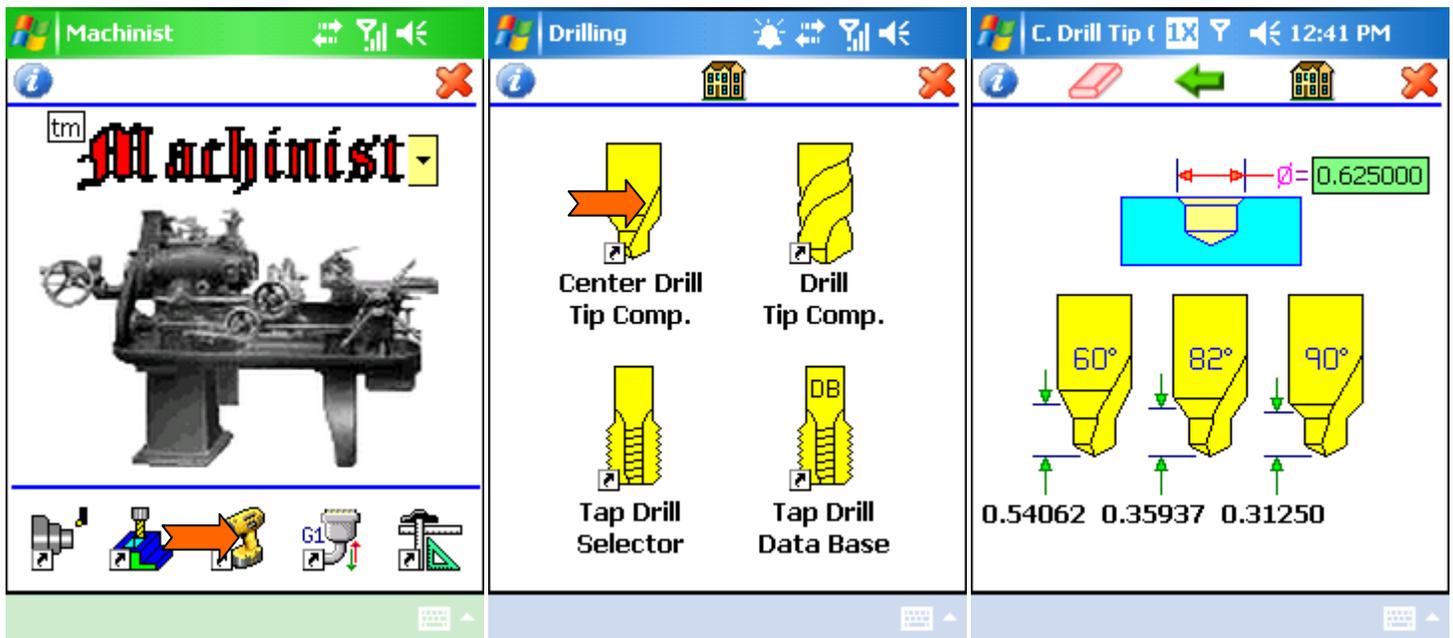
Tap  at any time to access the onboard step by step instructions.

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.

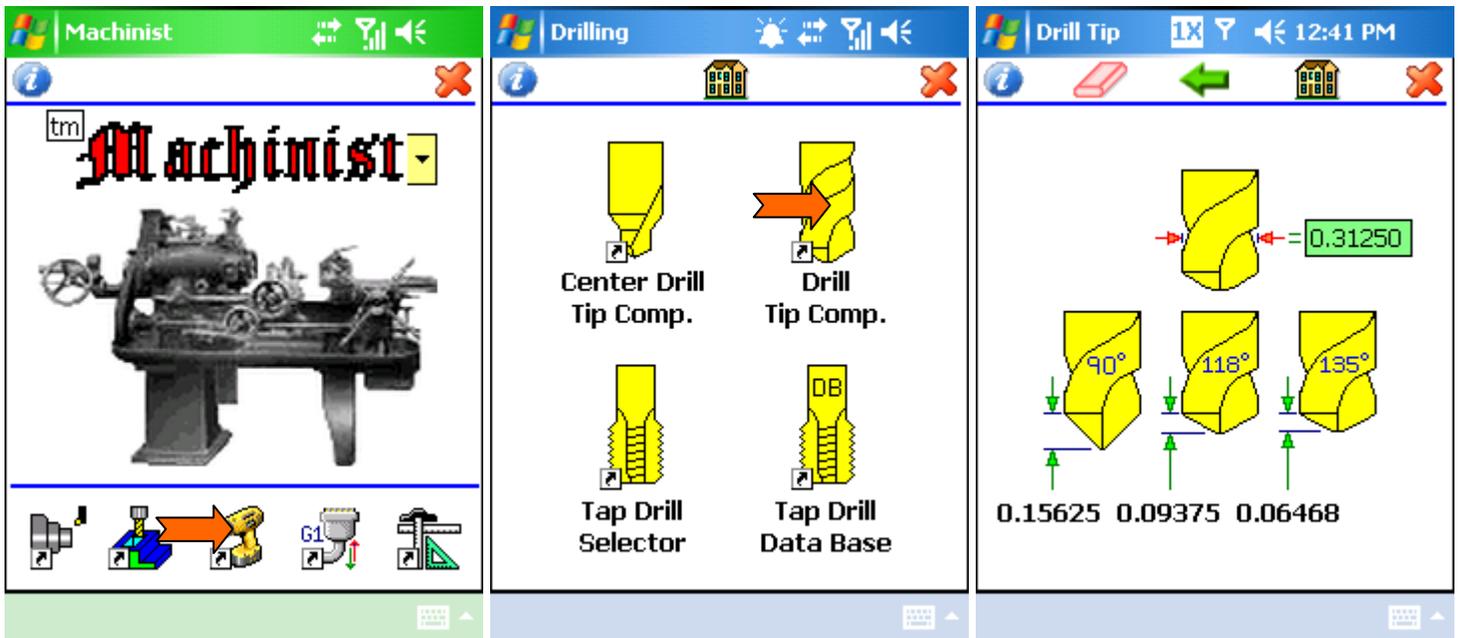
Tap  at any time to access the onboard step by step instructions.

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.

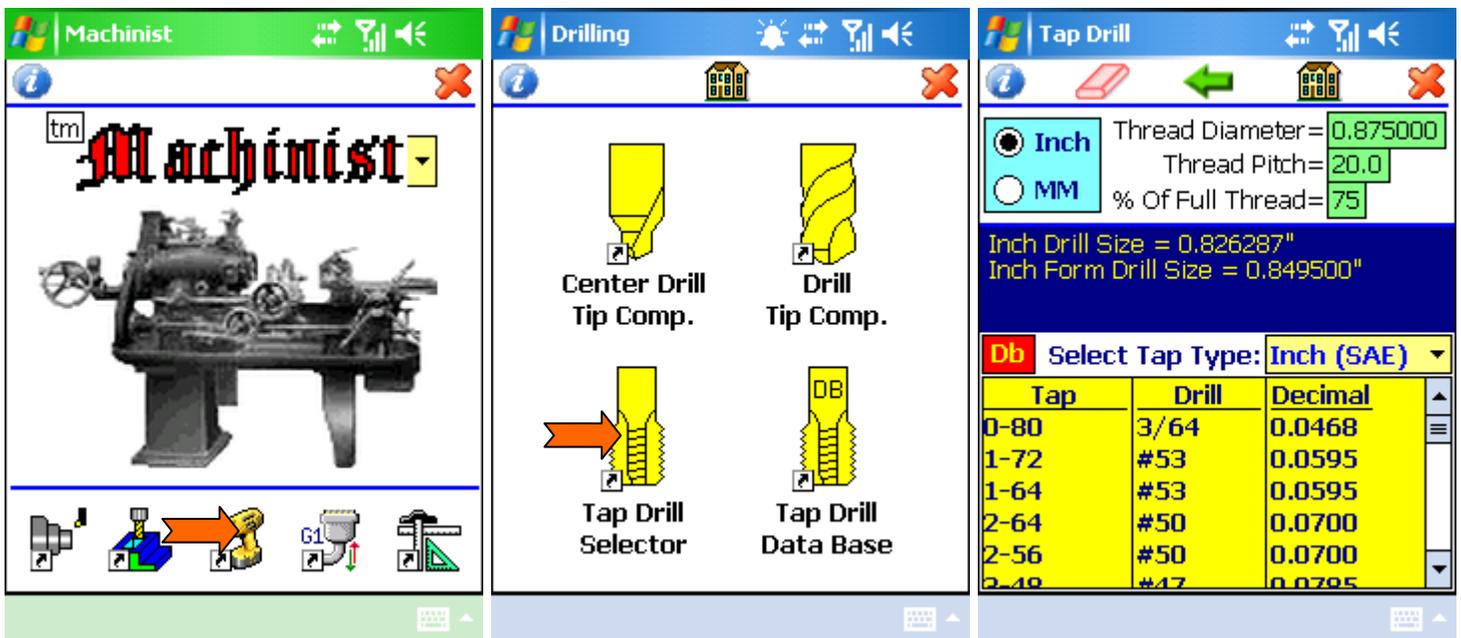
Tap  at any time to access the onboard step by step instructions.

Tap  to clear your information.

Tap  to return to Drilling main.

Tap  to return to **Machinist** main.

Tap  to exit **Machinist**.



The image displays three sequential screenshots of the Machinist app's Tap Drill Selector interface. The first screenshot shows the Machinist main menu with a large image of a lathe and a dropdown menu. The second screenshot shows the Drilling menu with four options: Center Drill Tip Comp., Drill Tip Comp., Tap Drill Selector (highlighted with a red arrow), and Tap Drill Data Base. The third screenshot shows the Tap Drill Selector screen with input fields for Thread Diameter (0.875000), Thread Pitch (20.0), and % Of Full Thread (75). It also displays calculated Inch Drill Size (0.826287") and Inch Form Drill Size (0.849500"). At the bottom, there is a table for selecting tap types.

Tap	Drill	Decimal
0-80	3/64	0.0468
1-72	#53	0.0595
1-64	#53	0.0595
2-64	#50	0.0700
2-56	#50	0.0700
2-49	#47	0.0795

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.**

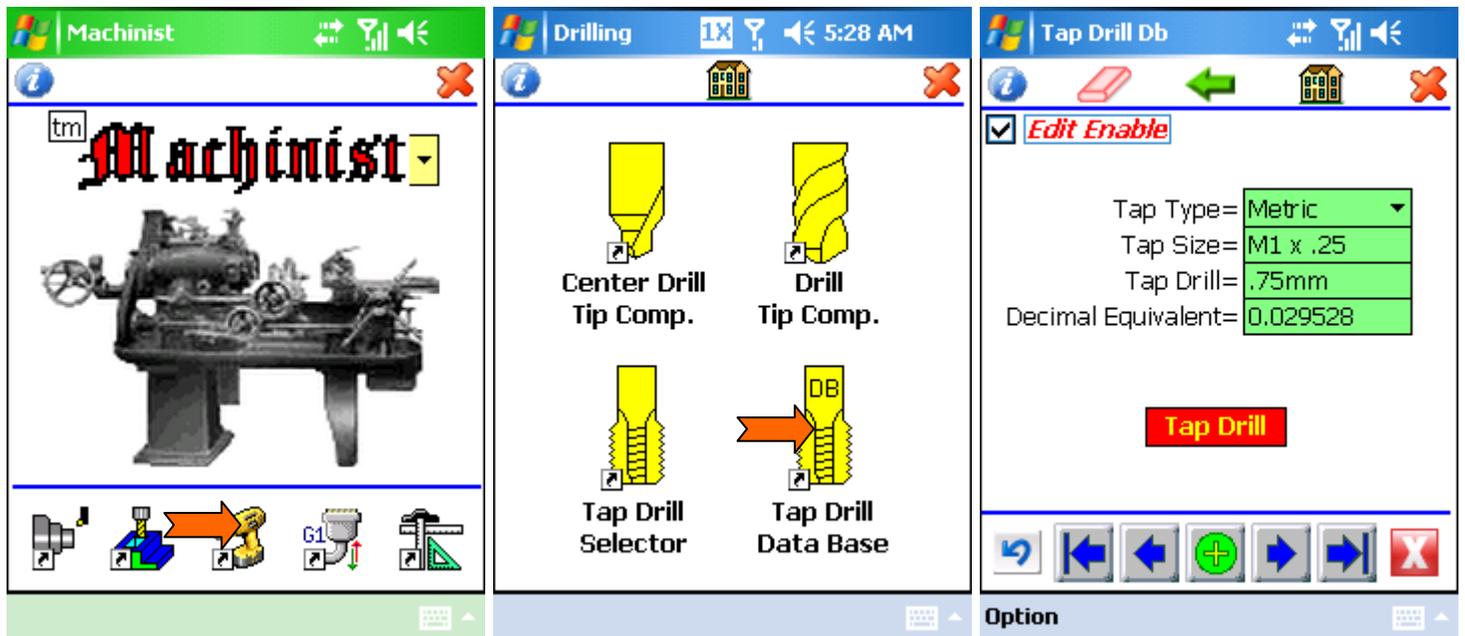
Tap  at any time to access the onboard step by step instructions.

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.

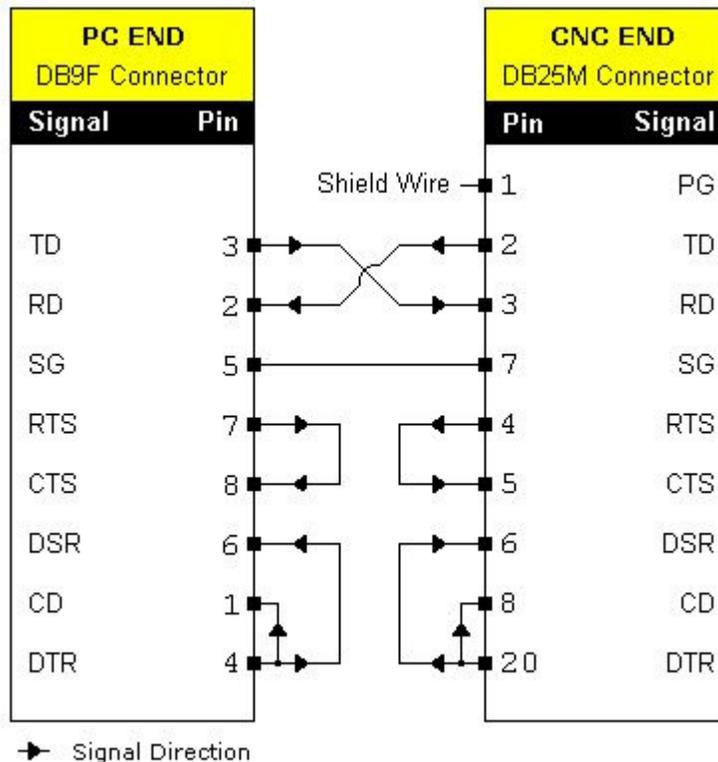
You must purchase or make a communication cable that will interface between the PocketPC and your CNC machine.

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).

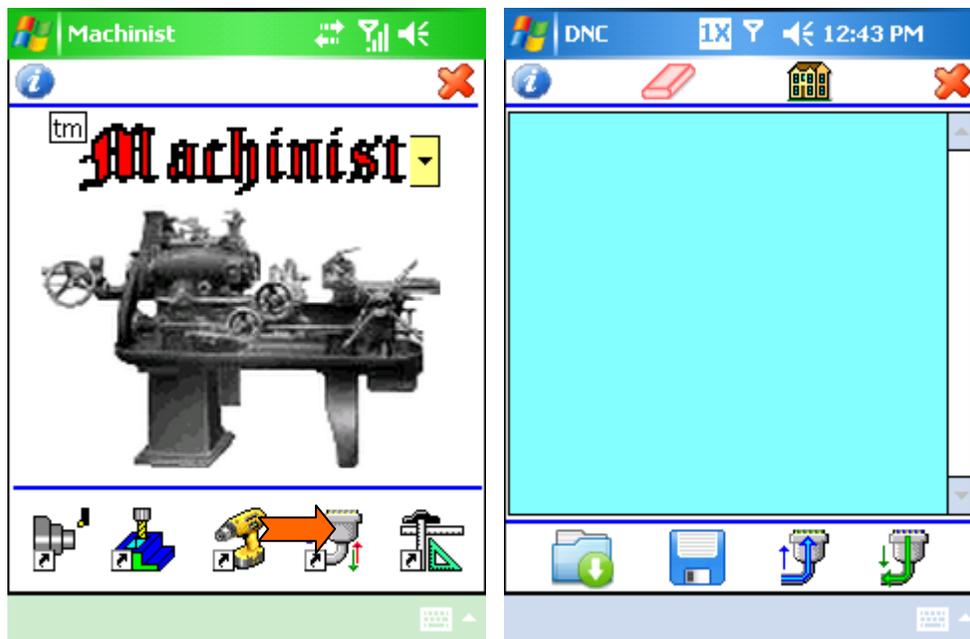
Tap  to access the onboard step by step instructions.

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.

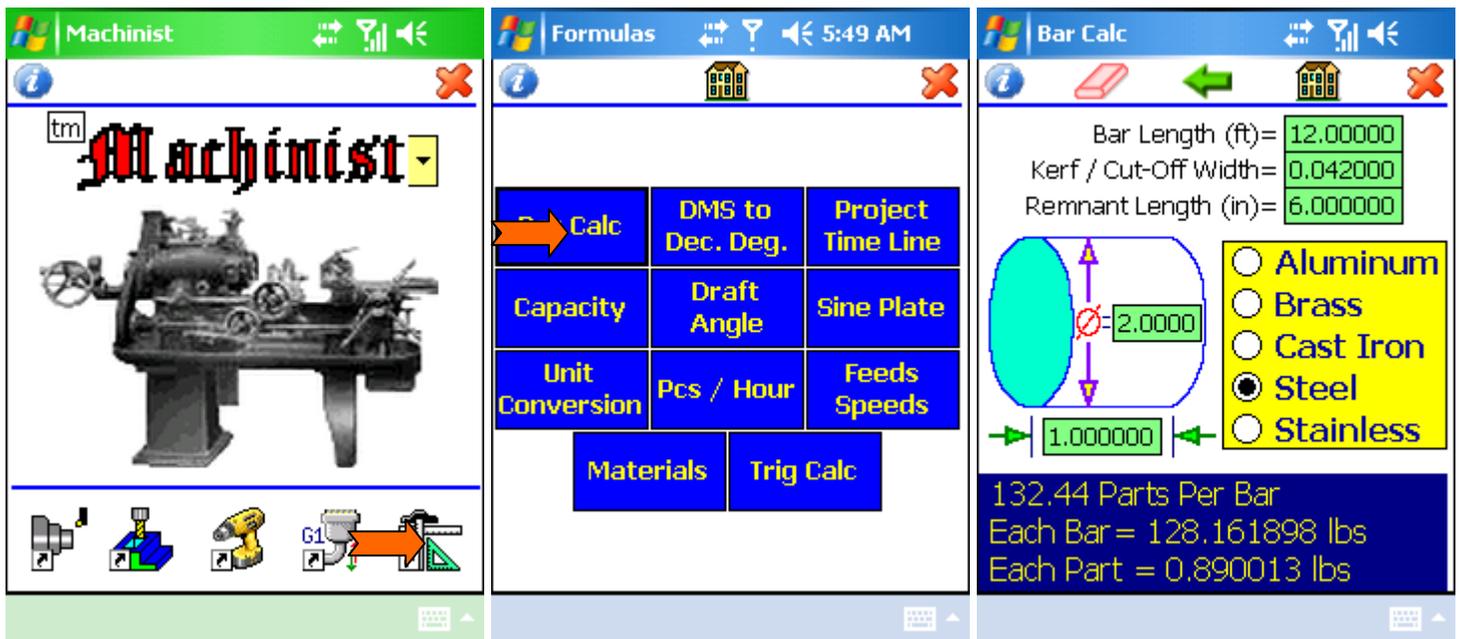
Tap  to access the onboard step by step instructions.

Tap  to clear your information.

Tap  to return to Formulas main.

Tap  to return to **Machinist** main.

Tap  to exit **Machinist**.



The image displays three screenshots of the Machinist app interface. The first screenshot shows the main menu with the 'Machinist' logo and a large image of a lathe. The second screenshot shows the 'Formulas' screen with a grid of calculation options: Calc, DMS to Dec. Deg., Project Time Line, Capacity, Draft Angle, Sine Plate, Unit Conversion, Pcs / Hour, Feeds Speeds, Materials, and Trig Calc. The third screenshot shows the 'Bar Calc' screen with input fields for Bar Length (ft) = 12.00000, Kerf / Cut-Off Width = 0.042000, and Remnant Length (in) = 6.000000. It also features a material selection list with radio buttons for Aluminum, Brass, Cast Iron, Steel (selected), and Stainless. A diagram shows a bar with a diameter of 2.0000 and a length of 1.000000. The results are: 132.44 Parts Per Bar, Each Bar = 128.161898 lbs, and Each Part = 0.890013 lbs.

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.

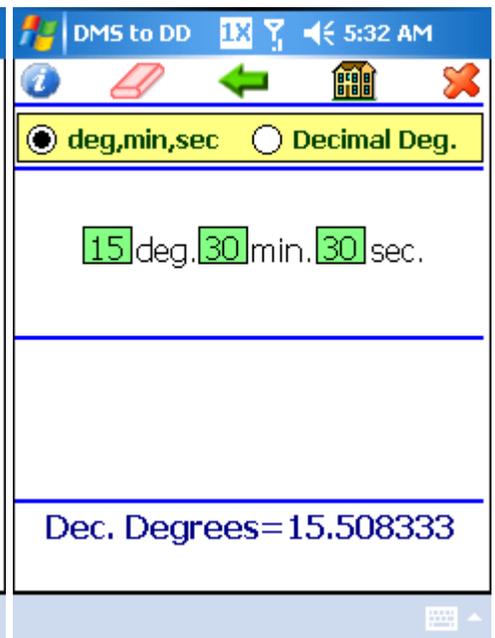
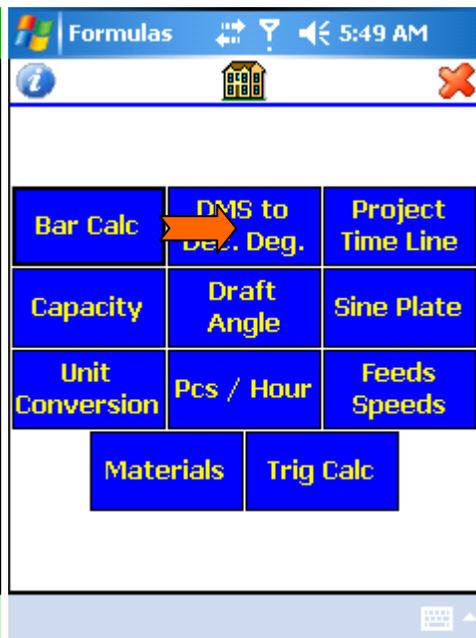
Tap  at any time to access the onboard step by step instructions.

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.

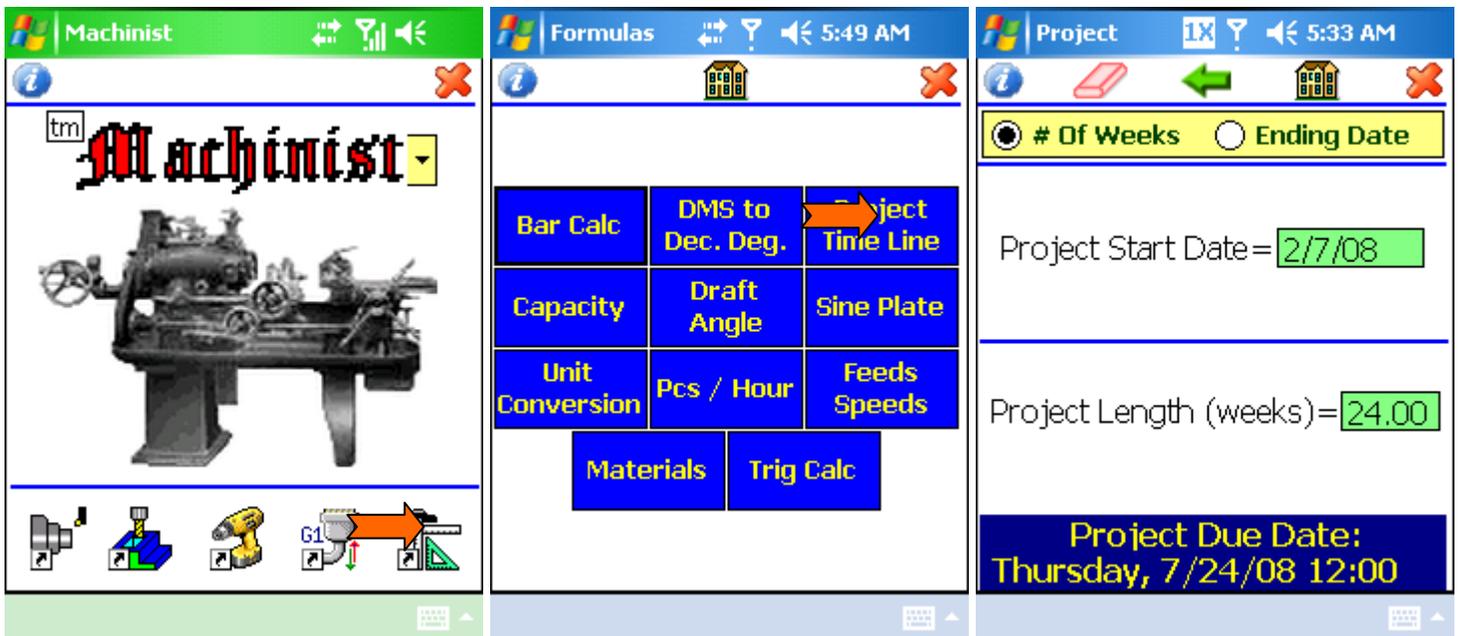
Tap  at any time to access the onboard step by step instructions.

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**.



The image displays three screenshots of the Machinist app interface:

- Left Screenshot (Machinist Main):** Shows the main menu with a green header, a home icon, a search icon, and a back icon. Below the header is a large image of a lathe machine and a dropdown menu labeled "Machinist". At the bottom, there are several icons representing different machining operations.
- Middle Screenshot (Formulas):** Shows the "Formulas" screen with a blue header, a home icon, a search icon, and a back icon. The screen displays a grid of buttons for various calculations: "Bar Calc", "DMS to Dec. Deg.", "Project Time Line", "Capacity" (highlighted with an orange arrow), "Draft Angle", "Sine Plate", "Unit Conversion", "Pcs / Hour", "Feeds Speeds", "Materials", and "Trig Calc".
- Right Screenshot (Capacity):** Shows the "Capacity" calculation screen with a blue header, a home icon, a search icon, a back icon, and an exit icon. The screen displays input fields for: Annual Requirement (pcs) = 375000.00, Work Weeks In A Year = 50, Quoted Pcs Per Hour = 50.00, Days Per Week Worked = 5, Hours Per Day Worked = 20.00, # Of Available Machines = 1.00000, and Required Efficiency = 100%. Below the input fields, a summary of results is displayed: 31250.00 Pcs Required / Month, 7500.00 Pcs Required / Week, 1500.00 Pcs Required / Day, 75.00 Pcs Required / Hour, 48.00 Sec. Required Cycle Time, and 1.50 Machines Required.

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.

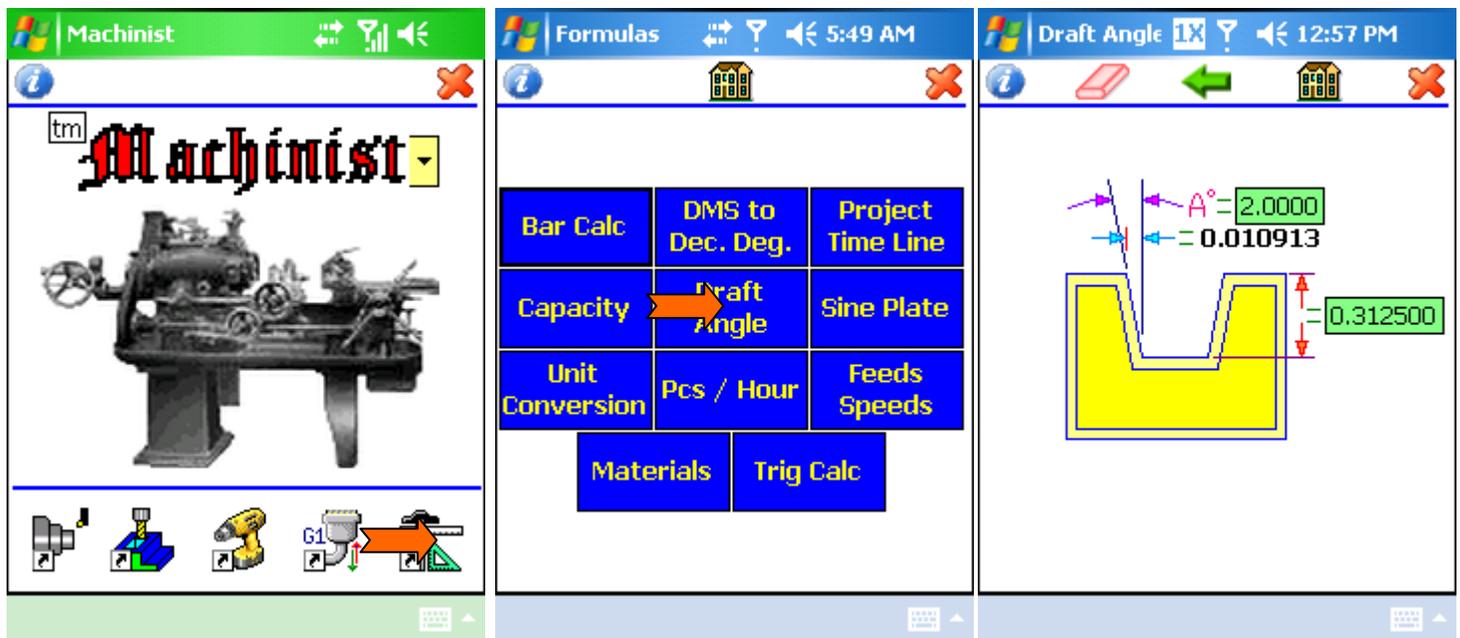
Tap  at any time to access the onboard step by step instructions.

Tap  to clear your information.

Tap  to return to Formulas main.

Tap  to return to **Machinist** main.

Tap  to exit **Machinist**.



The image displays three screenshots of the Machinist app interface:

- Left Screenshot (Machinist Main):** Shows the main menu with the title "Machinist" and a large image of a lathe. Below the image are several icons representing different machining tools and processes.
- Middle Screenshot (Formulas):** Shows a grid of buttons for various calculations. The "Draft Angle" button is highlighted with an orange arrow. Other buttons include "Bar Calc", "DMS to Dec. Deg.", "Project Time Line", "Capacity", "Sine Plate", "Unit Conversion", "Pcs / Hour", "Feeds Speeds", "Materials", and "Trig Calc".
- Right Screenshot (Draft Angle Calculation):** Shows the "Draft Angle" calculation screen. It features a diagram of a U-shaped part with a draft angle $A^\circ = 2.0000$ and a length of 0.010913 . The resulting draft angle is 0.312500 .

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.

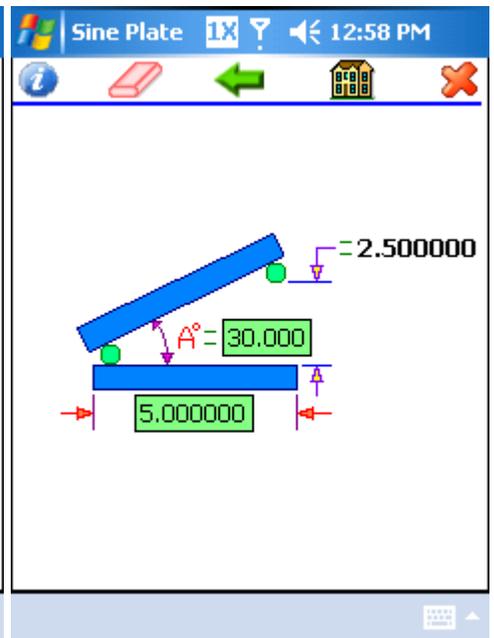
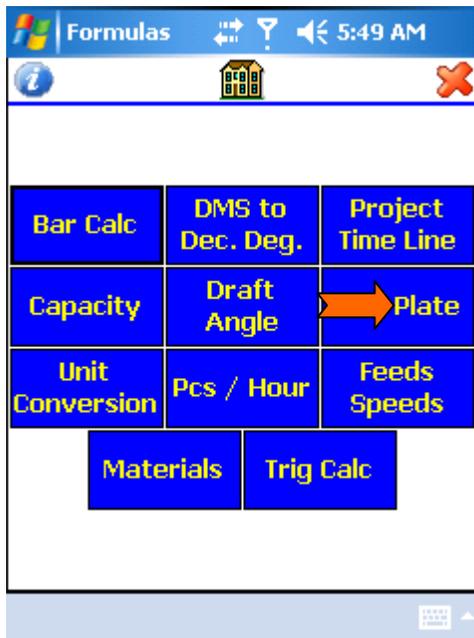
Tap  at any time to access the onboard step by step instructions.

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.

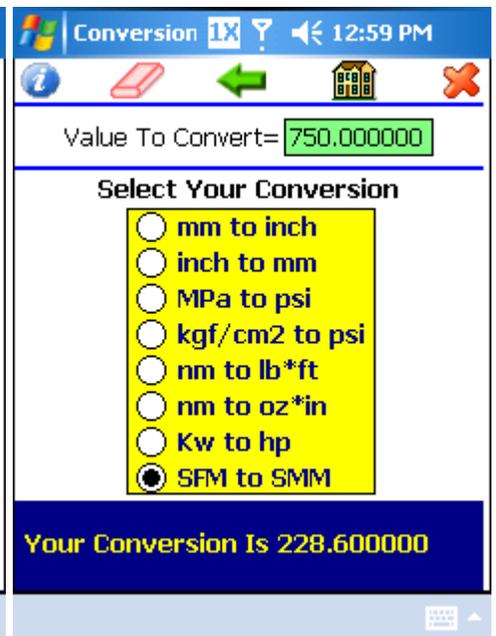
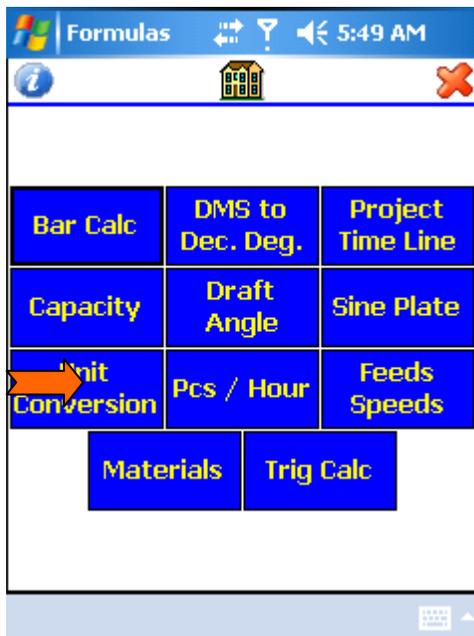
Tap  at any time to access the onboard step by step instructions.

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).

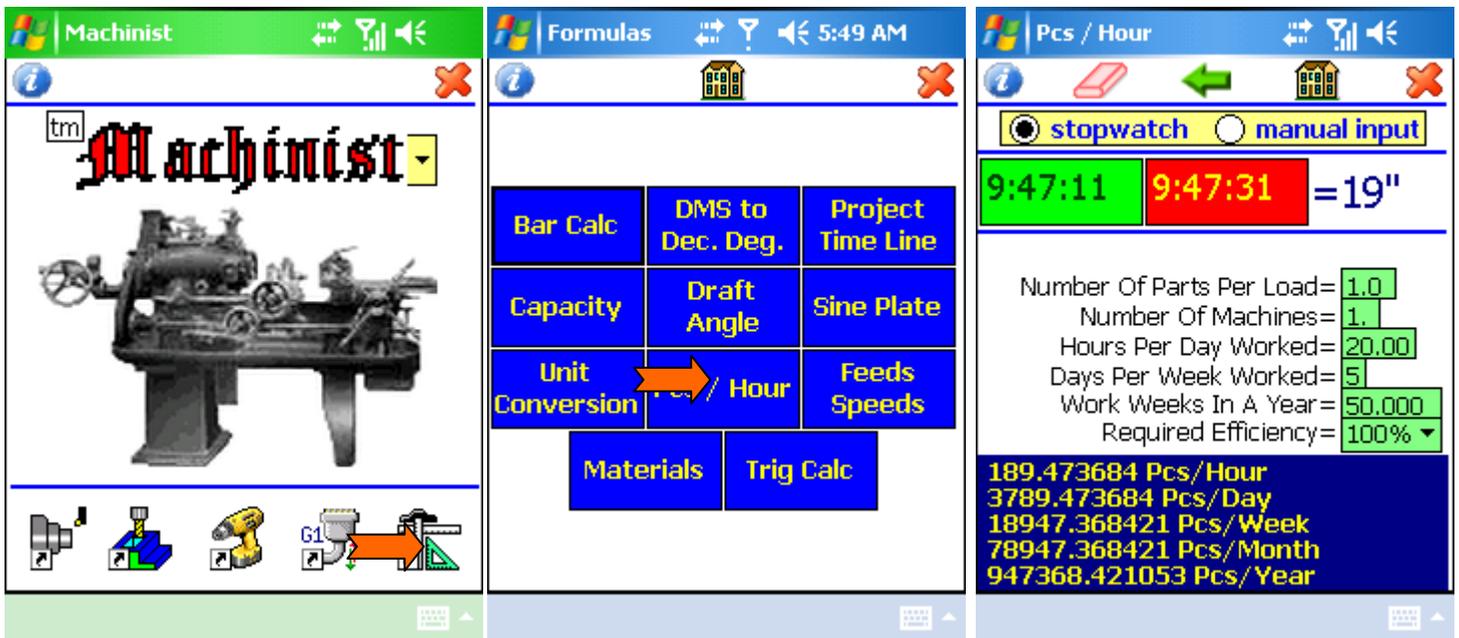
Tap  at any time to access the onboard step by step instructions.

Tap  to clear your information.

Tap  to return to Formulas main.

Tap  to return to **Machinist** main.

Tap  to exit **Machinist**.



The image displays three screenshots of the Machinist app interface:

- Left Screenshot (Machinist Main):** Shows the app title "Machinist" and a large image of a lathe. Below the image are several icons representing different calculation categories: Bar Calc, Capacity, Unit Conversion, DMS to Dec. Deg., Draft Angle, Materials, Project Time Line, Sine Plate, Feeds Speeds, and Trig Calc.
- Middle Screenshot (Formulas):** Shows a grid of calculation categories: Bar Calc, DMS to Dec. Deg., Project Time Line, Capacity, Draft Angle, Sine Plate, Unit Conversion, Feeds Speeds, Materials, and Trig Calc. An orange arrow points from "Unit Conversion" to "Pcs / Hour".
- Right Screenshot (Pcs / Hour):** Shows the "Pcs / Hour" calculation screen. It has a header with "stopwatch" (selected) and "manual input". Below the header, it shows a green box with "9:47:11" and a red box with "9:47:31" followed by "=19". Below this, there are input fields for: Number Of Parts Per Load=1.0, Number Of Machines=1, Hours Per Day Worked=20.00, Days Per Week Worked=5, Work Weeks In A Year=50.000, and Required Efficiency=100%. At the bottom, it displays the results: 189.473684 Pcs/Hour, 3789.473684 Pcs/Day, 18947.368421 Pcs/Week, 78947.368421 Pcs/Month, and 947368.421053 Pcs/Year.

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 IPR (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).

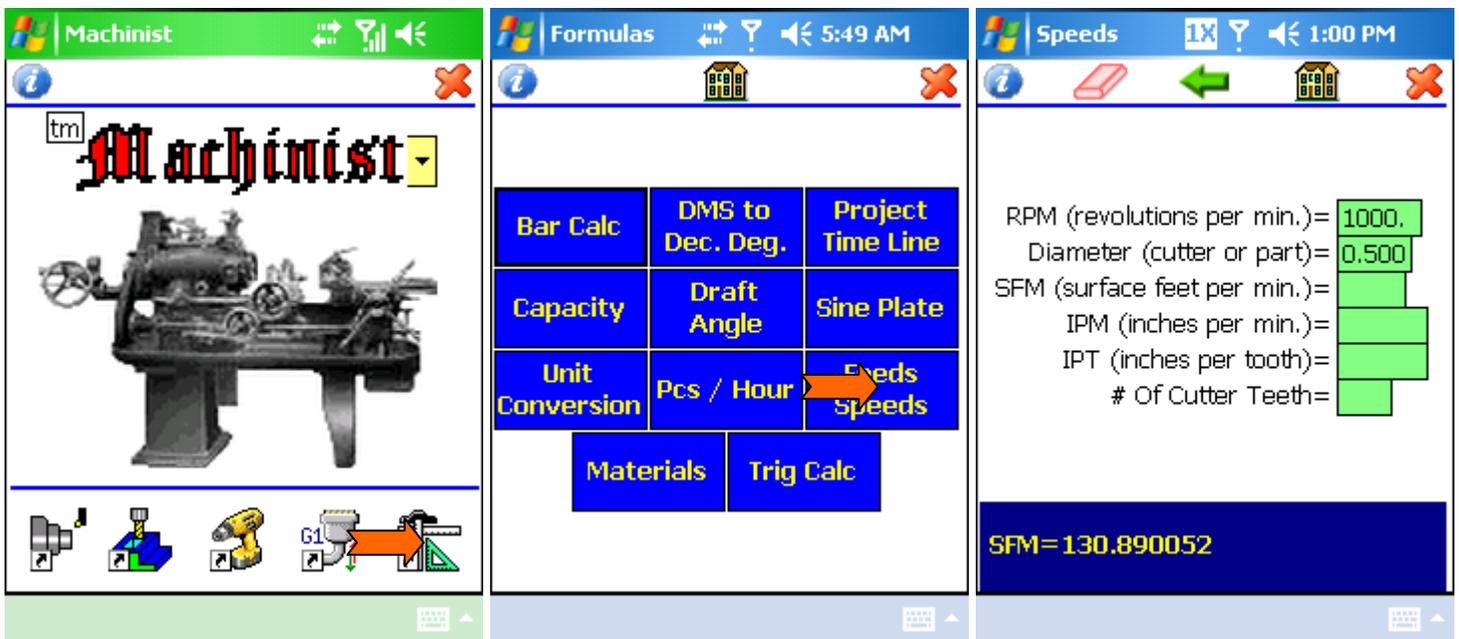
Tap  at any time to access the onboard step by step instructions.

Tap  to clear your information.

Tap  to return to Formulas main.

Tap  to return to **Machinist** main.

Tap  to exit **Machinist**.



The image displays three screenshots of the Machinist app interface. The first screenshot shows the main menu with the 'Machinist' logo and a large image of a lathe. The second screenshot shows the 'Formulas' menu with options like 'Bar Calc', 'DMS to Dec. Deg.', 'Project Time Line', 'Capacity', 'Draft Angle', 'Sine Plate', 'Unit Conversion', 'Pcs / Hour', 'Feeds Speeds', 'Materials', and 'Trig Calc'. The third screenshot shows the 'Feeds-Speeds' calculation screen with input fields for RPM (1000), Diameter (0.500), SFM, IPM, IPT, and # Of Cutter Teeth, and a result for SFM = 130.890052.

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).

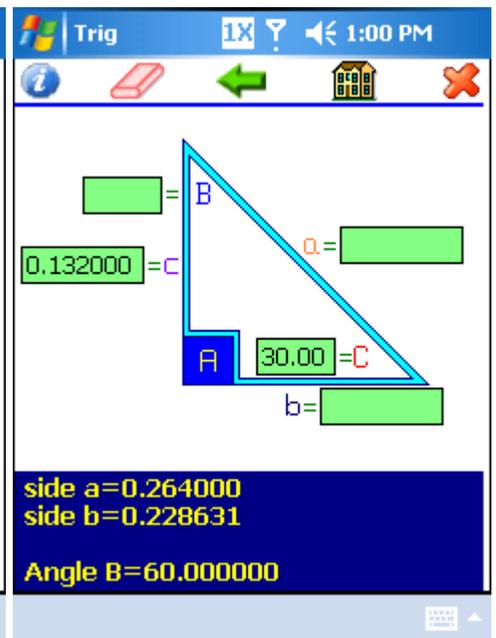
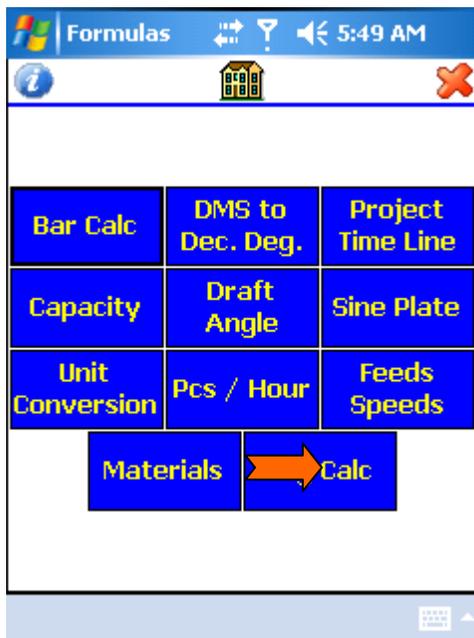
Tap  at any time to access the onboard step by step instructions.

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**.