Mach3 2010
Reference Manual
copyright © 2010-2012 Gerry Grzadzinski. All Rights Reserved

By installing this software, the user is agreeing to the license agreement found in
2010_License.txt
Please read before installing.
A copy of this manual and the 2010 Probing manual will be installed in the \Mach3\Documentation folder
for your convenience. Shortcuts to the manuals can be found in the Start menu as well.

Installation:

Due to a change in installers to facilitate multiple language versions, if a previous version of the 2010
Screenset has been installed (including the demo), I recommend uninstalling it prior to installing this
version.

Run Install-2010-screenset-1.1.4.exe, choose a language (English or French), and choose the location of
your Mach3 installation folder. This is typically C:\Mach3. In most cases, the installer will detect the
location for you. Note: if you use the Browse button to choose a different location, a bug in the installer
may add a duplicate folder name to the end of the path. Check the returned path carefully and edit as
needed before clicking Next>.
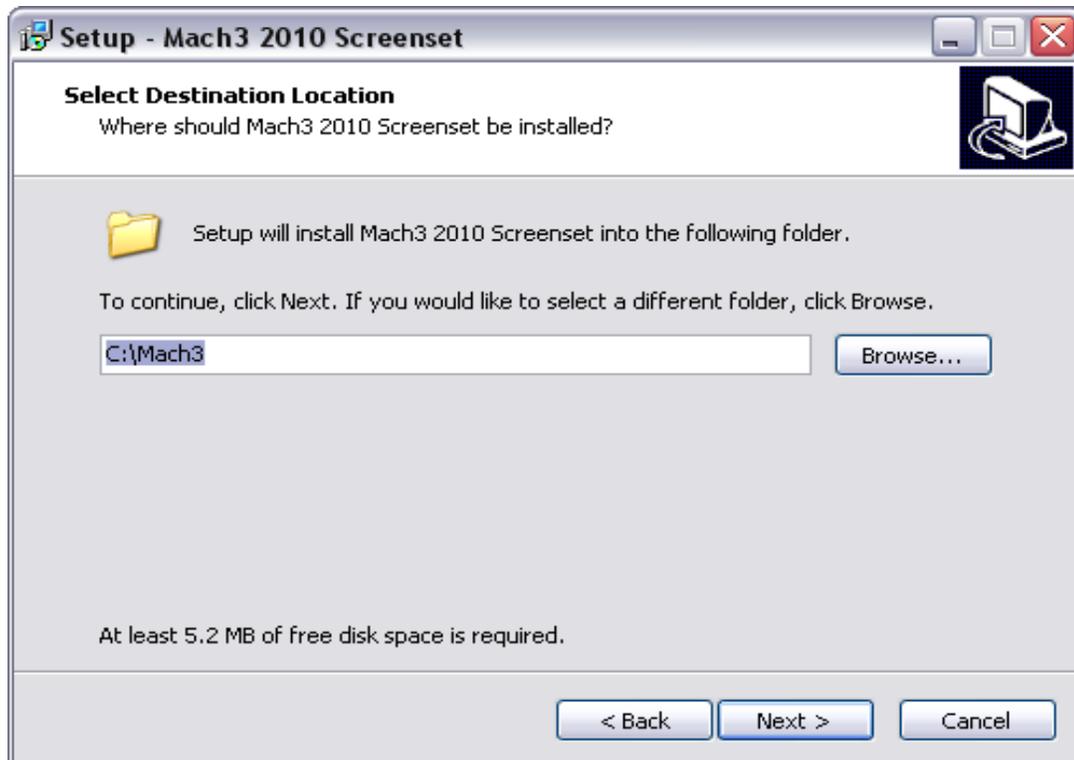
Note: The 2010 Screenset can be uninstalled by using it's Start Menu shortcut (Mach3 2010 Screenset), or by using
Add/Remove programs.

<u>Copy Macros</u>
Once the instalation is finished, several macros need to be manually installed. The 2010 Screenset macros are installed by default into \Mach3\macros\2010. They need to be copied into \Mach3\macros\Your_Profile. If you don't know the profile name you're using, it can be found in Mach3 when running the default 1024 Screenset, in the lower right corner. Before copying the 2010 macros, you should backup the existing <u>M6Start.m1s</u> and <u>M6End.m1s</u> macro's, in case you decide to remove the 2010 screenset. Then copy the following macros from \Mach3\macros\2010.

M6Start.m1s
M6End.m1s
M881.m1s   (Initial Auto Zero)
M882.m1s   (Go To Toolchange Position)
M883.m1s   (Go To Park Position)
M884.m1s   (Ref All)
M885.m1s   (Ref X)
M886.m1s   (Ref Y)
M887.m1s   (Ref Z)
M888.m1s   (Ref A)
M889.m1s   (Simple Auto Zero)
M890.m1s   (Zero to Laser Crosshair)
M901.m1s   (X+ Probing)
M902.m1s   (X- Probing)
M903.m1s   (Y+ Probing)
M904.m1s   (Y- Probing)
M905.m1s   (X+Y+ Probing)
M906.m1s   (X+Y- Probing)
M907.m1s   (X-Y+ Probing)
M908.m1s   (X-Y- Probing)
M909.m1s   (Circular and Rectangular Pocket Probing)
M910.m1s   (Outside Cylinder Probing) New 11/8/11
M911.m1s   (X Pocket Probing)
M912.m1s   (Y Pocket Probing)

Load ScreenSet
Once the macros are copied, you can load the Screenset in Mach3, by using the "Load Screens" option in the View menu. For 3 axis machines, choose 2010.set, which is found in the main Mach3 folder. Three additional screensets are provided.

3 Axis (Standard 2010 Screenset) –   2010.set
3 Axis w/ extended Diagnostics –   2010x.set
4 Axis version –   2010-4.set
4 Axis w/ extended Diagnostics –   2010-4x.set

You should now be looking at the 2010 Screenset.

For best results, the 2010 Screenset MUST be run on a 1024x768 monitor (with taskbar set to Auto-Hide), or, on a larger monitor, in a 1024x768 window.
Running at other resolutions will result in blurry or missing text, and other graphics.
In General Config, both "Auto Screen Enlarge" and "Boxed DRO's and Graphics" should be left unchecked.

Two widescreen versions of the 2010 Screenset have been graciously donated by user Kyel Smith. They are:
2010 (1680x1050).set and 2010 (1920x1080).set
New 4/13/12
4 axis versions of these are 2010 (1680x1050)-4.set and 2010 (1920x1080)-4.set

Setup

All DRO's that contain user entered values are saved when Mach3 is closed. It is recommended to close Mach3 and restart after entering these values, to prevent them from being lost and needing to be re-entered.

Brains
The 2010 Screenset requires two brains which operate functions of the screen. These need to be enabled for the screen to function properly.

Go to Operator>Brain Control, and enable the following two brains.
Axis_Scale.brn
Offset LEDs.brn
This is done by selecting each brain individually, and checking the "Enabled" box. When both are enabled, click OK.

Auto Zero Setup
See the Auto Zero section for Auto Zero setup information. (pages 9-14)
Warning!! Failure to properly set up the DRO's and Check box's on the Toolchange tab can result in damage to the machine and or broken tools. The screenset has several safety checks to prevent potential unwanted movement, but relies on user information, which has the potential to cause damage if not set correctly.

Park Position
Set the Park Position on the Offsets tab.
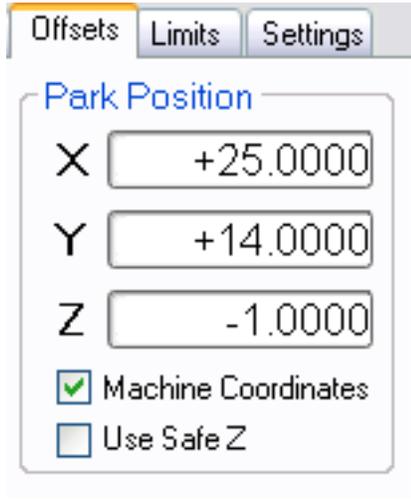
This is a user definable position where the machine can be moved to with the push of a button (or Control-P).

Checking the "Machine Coordinates" checkbox toggles whether the position is in Work coordinates or Machine coordinates.

Checking "Use Safe Z" uses the value of SafeZ during the move if it's enabled.

Due to a limitation in Mach3, whether or not SafeZ for the Park position is in Work or Machine Coordinates is determined by the Machine Coordinates Checkbox above.

Jog Settings
Set the Jog Increments on the Settings tab. The current value being used can be cycled up and down on the Jog tab.

Diagnostics Screen
To position the Diagnostics screen, press the tab key 3 times, once to open it, once to close it, and again to open it. Now drag the left edge until it is fully opened. Now close it with the tab key, or the X in the upper right corner. Close and restart Mach3 to save it's position.

Toolbar Buttons - **Shortcut Keys**

Open G-code file   -   Control-O

Edit G-code file   -   Control-E

Close G-code File   -   Control-C

Select Wizard   -   Control -W

Mach3 Mill Add-Ons Wizard
Loads Mach3 Mill Add-Ons Wizard (Newfangled Solutions)

Toolpath On/Off   (On State)
Toggles toolpath display on/off.

Regen Toolpath

Open G-Code List
Opens Mach3MillGcode.htm in a browser window

XY Probing Wizard
Loads 2010 Probing Wizard

Simple Auto Zero Macro
Executes the Simple Auto Zero macro, M889.m1s

Initial Auto Zero Macro for Tool Changes
Executes the Initial Auto Zero macro, M881.m1s

Go To Toolchange Position
Rapid move to the Toolchange Position as specified in the DRO's on the Toolchange Tab. The Toolchange Position is in MACHINE COORDINATES. Z moves first, followed by a combined XY move. If SafeZ is enabled in Config>SafeZ, then a SafeZ move is done first, followed by an XY, followed by a Z move to the Tool Change Position. (M882.m1s)

M6 Macro
Executes the M6 toolchange macros. Same as M6 in MDI or g-code.

Mist On/Off
Toggles Mist coolant On/Off. Same as M7 in MDI or g-code. Can also be turned off with M9.

Flood On/Off
Toggles Flood coolant On/Off. Same as M8 in MDI or g-code. Can also be turned off with M9.

Spindle On/Off
Toggles spindle On/Off. Same as M3 in MDI or g-code. Can also be turned off with M5.

Zero to Laser Crosshair
Sets X and Y position DRO's to 0,0 at position of Laser Crosshair. (M890.m1s) See page 14 for setup information.

Ref All Home - Control-Home
Executes VB Homing script. Script may be edited by using "VB Script Editor" from the Operator menu. (M884.m1s)

Ref X
Executes VB Homing script for X axis. Script may be edited by using "VB Script Editor" from the Operator menu. (M885.m1s)

Ref Y
Executes VB Homing script for Y axis. Script may be edited by using "VB Script Editor" from the Operator menu. (M886.m1s)

Ref Z
Executes VB Homing script for Z axis. Script may be edited by using "VB Script Editor" from the Operator menu. (M887.m1s)

Ref A
4 axis screens only. Executes VB Homing script for A axis. Script may be edited by using "VB Script Editor" from the Operator menu. (M888.m1s)

De-Ref All Axis

Go To Safe Z - Control-Z
Rapid move to Safe Z position if SafeZ is enabled.

GoTo Zero's - Alt-Control-Z
Rapid Move to 0,0,0

Go To Park Position - Control-P
Rapid move to "Park" Position as specified on "Offsets" Tab, using "SafeZ" and "Machine Coordinates" options if enabled.
Note: Safe Z for Park Position moves will use coordinate system specified in Park Position options, which may or may not be the same coordinate system used for normal SafeZ operation. (M883.m1s)

Diagnostics - TAB
Opens slide out Diagnostics screen.

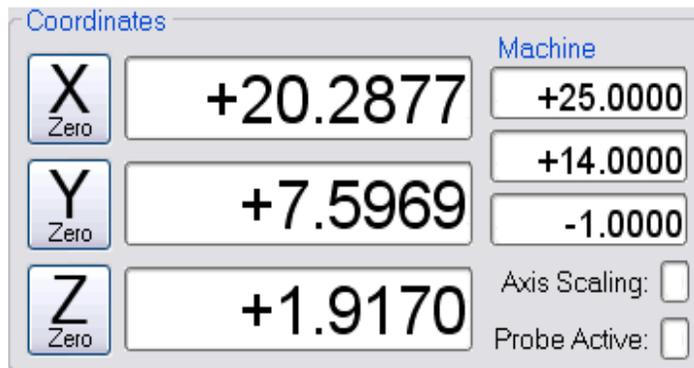**Reset** - ` (The key with the ~, same as default Mach3)
Note: The Reset LED is located in the lower right corner of the screen.

Using the 2010 Screenset

It is assumed the user has a working knowledge of the Mach3 CNC Control Program. Standard Mach3 functions are not covered in this manual

Main DRO area

Current X, Y, and Z coordinates are displayed in large DRO's. Buttons to the left of the DRO's will reset their values to zero. When an axis is Referenced (Homed), it's corresponding zero button will have a green border.


Machine coordinates are always displayed in a set of smaller DRO's to the right of the Work coordinates.

Axis Scaling: This is a "global" LED that indicates if any axis' have scaling applied. Individual axis scale DRO's are located in the Options tab.

Probe Active: LED to indicate when the Auto Zero probe plates are active. Useful for testing prior to running Auto Zero routines.

Jogging tab
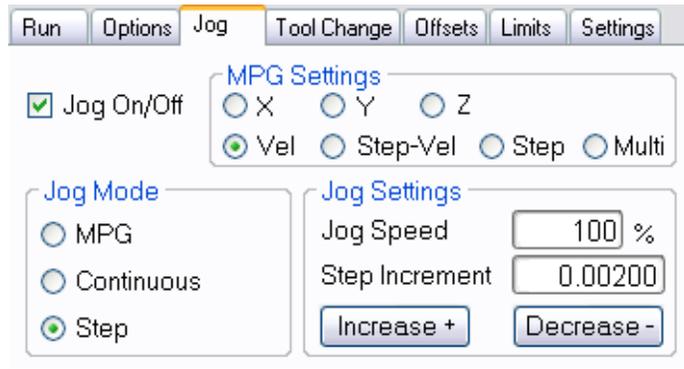
There are no "Jog" buttons in the 2010 Screenset. Jogging is done with the keyboard, with keys assigned using Config>Hotkeys.

Jog mode can be selected by clicking the desired option, or cycled using the "End" key.

Jog Speed and Step Increment values can be increased with the "+" key on the numeric keypad, or decreased with the "-" key on the keypad. If in Continuous Jog mode, then the Speed% is changed. If in Step Jog mode, then the Increment is changed. The values used are user defined on the Settings tab.

Tabs
Each Tab can be selected with the following shortcut keys:
Run -   F1
Options -   F2
Jogging -   F3
Tool Change - F4
Offsets -   F5
Limits -   F6
Settings -   F7

In addition, the F12 key will cycle through the different tabs.

Auto Zero Routines

The 2010 Screen set includes both a "simple" Auto Zero and a modified version of the macros created by Big Tex at CNC Zone. http://www.cnczone.com/forums/mach_wizards_macros_addons/100594-mach_3_tool_setter_big-tex_version.html

The following information applies to the Auto Zero for use with tool changes. See the end of this section for more info on the Simple auto zero.

Auto Zero Requirements:

In order to use these macros, the machine must be referenced to a repeatable position prior to running the macros. This is typically done with home switches, but you can also reference to hard stops if you don't have home switches. In Mach3's Config>Homing Limits screen, the Z axis Home Offset must be set to 0, and Auto Zero should be enabled. This will ensure that Z zero in machine cordinates is the highest the Z axis can travel. **The -Z direction must be DOWN.**

See the note below if your machine is not equipped with a Z axis home switch.
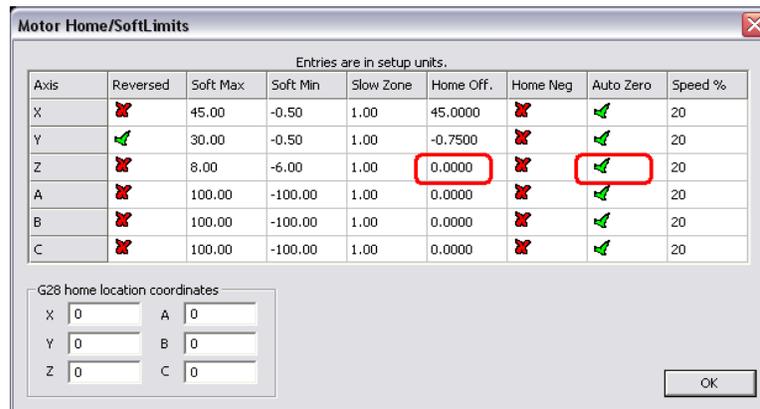
The machine must also have two reference plates connected to Mach3's probe input, which are activated when the the tool comes in contact with them. One plate must be moveable, and able to be placed under the tool atop the workpiece, and the other plate should be permanently mounted to the machine, and accessible by the tool in the spindle. The location of the fixed plate is specified in the Toolchange tab.

In General Config, Toolchange, select "Stop Spindle, Wait for Cycle Start".

**Using Without Z Home Switch: It hasn't been tested, but it should be possible to use this without a Z axis home switch. To do so, edit the homing script used so that it only home's X and Y. Jog the Z axis to it's highest position, and Ref (Home) the Z axis. This should set the Z axis Machine Coordinates to 0.0.
If the "Persistent DRO's" option is selected in General Config, then this should only need to be done once, provided the Z axis never loses position. It might be a good idea to periodically re-Ref the Z, or at the very least visually verify it each time you intend to use the Auto Zero.
The following settings on the Toolchange tab **<u>MUST</u>** be set prior to running the Auto Zero routines.

Change Position.

X Pos.: X MACHINE COORDINATE of Toolchange position
Y Pos.: Y MACHINE COORDINATE of Toolchange position
Z Pos.: Z MACHINE COORDINATE of Toolchange position
When an M6 is encountered in the g-code, or the M6 button is pressed, the machine will travel to these coordinates and wait for the user to change the tool**.** The Toolchange Z Position is used for the rapid height of the move to the tool change position. Z moves first, followed by an XY move.
Note: If the Clearance Plane below is above the Toolchange Z position, the XY move will be at the Clearance Plane height.

Ignore M6 Calls Checkbox. NEW
When checked, no toolchanges will occur when an M6 is encountered. The machine will stop and wait for Cycle Start, but no other functions of the M6 macros will take place.
This is useful when running g-code programs that use a single tool, but still contain an M6 call at the start. Note that the tool will need to be zeroed with either the Simple Auto Zero, or the Initial Auto Zero buttons, as the M6 will not zero the tool.

Auto Zero Settings.

Plate Thickness: Thickness of moveable reference plate.

Fixed Plate X: X MACHINE COORDINATE of fixed reference plate.

Fixed Plate Y: Y MACHINE COORDINATE of fixed reference plate.

Clearance Plane: Rapid Z level used during auto zero macro's, in WORK COORDINATES. After Z zero is set, the tool will move to this Z location.
Warning! The Z Clearance Plane is designed to be set once and left alone. It's sole purpose is to increase the speed of the auto-zero routines by choosing a safe Z position, rather than moving to Z Machine zero each time. It should be set one of the following two ways, based on the Z location of the top of your fixed plate. Note: The following examples assume that Z zero is the top of stock.

1) If your fixed plate is above the bed or table of the machine, the Clearance Plane should be set at minimum, to the distance from the bed to the top of the fixed plate. For example, if the fixed plate is 1" (25mm) thick, and bolted to the bed or table, then the Clearance Plane should be set to at least approx. 1-1/8", or 28mm. Setting it lower can result in the tool crashing into the fixed plate!!
2) If your fixed plate is level with or below the bed or table of the machine, the Clearance Plane should be set at a minimum, to a distance greater than the moveable plate thickness. Setting it lower will result in the tool plunging into the plate after auto-zeroing!! Note that if Z zero is set to the bottom of the workpiece, then the Clearance Plane should be set to a value greater than the material thickness.

Material Offset: Use the Material Offset option to zero the Z axis to a position offset from the plate bottom. Check the box to enable this option. If unchecked, the offset value is ignored.

Example: Say that you have g-code that cuts profiles .74" deep into a .75" thick panel, and you'd like to use the same code on thinner material, say .50". There are two ways to use the offset.
1) Set the offset to the difference between the material originally coded for (.75") and the material your using (.5"). You'd set the offset to .25", and zero with the plate on the material. Z zero will be set .25" above the material.
2) Set the offset to the thickness of the original material, .75", and zero to the bed of the machine. Z zero will be set .75" above the bed. This will allow you to cut the profiles from any material thickness not greater than the original.
Note: Negative values are allowed for the material offset. If a negative value is used, a message will be displayed to notify the user, followed by a 3 second pause, before the probing operation begins.

Manual Spindle Control: Check this box if you **Do Not** have Mach3 turn your spindle On/Off. Enabling this option causes Mach3 to stop and wait for you to start your spindle before resuming.

What Not To Do!!!

Many safety checks have been included in these auto zero routines to prevent machine crashes, both due to improper user input, or even machine errors. However, due to the numerous possible Mach3 configurations, it's impossible to protect against all possible errors.

Below is a list of known issues which can possibly cause a tool crash, either into the workpiece, or machine itself.

1) Current units (G20 or G21) must be the same as the machine's Native Units (Config>Native Units). Example: if your machine is set up in inches, and you run code with a G21 in it, the machine **WILL** crash during the auto zero.
2) Using G92 offsets will very likely cause the machine to crash during the auto zero.
3) Using G43 tool length offsets will very likely cause the machine to crash during the auto zero.
4) Setting the clearance plane (on toolchange tab) below the fixed plate **WILL** result in the tool crashing into the fixed plate.

Auto Zero Operation

Prior to running gcode, the first step should be mounting the workpiece to the machine.

Make sure the spindle is OFF!!

Note: It's a good idea to touch the plate to the tool and verify the plate is functioning properly via the on screen LED prior to starting.

Place the moveable plate on the workpiece, and jog the tool over the plate. The closer it is to the plate, the quicker the operation will be.

Press the Initial Auto Zero button.

The tool will begin moving downward at 10ipm (250mm/min) until it reaches the plate. It will then retract .05" (1mm) and probe again at 1ipm (25mm/min) to get a more accurate reading.

Z zero is then set to the top of the workpiece.

The tool will then retract to the value specified in the Clearance Plane DRO, and rapid to the fixed plate position, where it will repeat the probing process to get a reference position for future tool changes. Once finished, the tool will retract back to the Clearance Plane, and rapid back to it's original XY position. At this point you can load your g-code and start cutting.

Note: If a g-code file has been previously loaded, then the machine will immediately begin executing the g-code. The g-code MUST contain an M3 to start the spindle, unless the "Manual Spindle Control" option is selected.

Tool Change Auto Zero Operation

Note: The Initial Auto Zero routine **<u>MUST</u>** be performed prior to an M6 being called, or Z zero will be set incorrectly. This may result in damage to both the workpiece and the machine!!

When Mach3 encounters an M6 in your g-code, the spindle will turn off, and the tool will travel to the Toolchange Position, and wait for the user to change the tool. Mach3 will prompt the user to press Cycle Start. After changing the tool, press Cycle Start. The tool will rapid to the fixed plate position, and begin the probing operation as desribed above. Using the previously calculated offset, Z zero will be set to the top of the workpiece. The tool will then retract to the Clearance Plane, and rapid back to it's original XY position prior to the M6 being called, and cutting will resume.

Note: For safety reasons, the M6 macro will NOT restart the spindle after a toolchange. The G-Code should contain an M3 on the line following the M6, to re-start the spindle when cutting resumes.
Example:
N050 M6T2
N051 M3
......
......

If "Manual Spindle Control" is selected, the machine will wait for you to turn on the spindle before resuming.

Limitations:
If you have an M6 in your g-code for the first tool, Mach3 will run the auto zero, even though you have previously ran the Initial Auto Zero on this same tool. There are 4 possible workrounds for this.
1) Edit the CAM programs post processor to not use an M6 for the first tool.
2) Hand edit the g-code to remove the M6.
3) Let the machine Auto Zero again. If time is not an issue, this of course is the easiest option.
4) New Check "Ignore M6 Calls" checkbox. Useful if g-code only uses one tool, but still contains an M6.

Manual Use:
If the user prefers to not use M6 in their g-code, and instead create separate g-code programs for each tool, then use the following procedure:

1) Run Initial AutoZero macro, and run the first g-code program. Close the g-code file.
2) Use the M6 button on the toolbar to change tools and auto zero the new tool. If you load the next g-code prior to the M6, program execution will begin immediately following the auto zero routine.

Note: Due to the way that Mach3 handles the M6 macro's, if an error occurs during the auto zero process, Mach3 must be put into an E-Stop condition to abort the tool change. An error message will notify the user if this occurs.

There is one exception to this. The macro checks the Clearance Plane against Machine Z zero. If the Clearance Plane is above the Z Machine zero (typically the home switch, if installed), the user will get the option of aborting the macro, or continuing with the Clearance Plane reset to .125" or 2mm below Machine Z zero. This value can be modified in the M6end macro. It's assigned to the variable ClearAllow.

Simple Auto Zero

For times when no toolchanges are required, the "simple" auto zero is a much quicker option for zeroing the tool to the work.

To use the simple auto zero, jog the tool over the touch plate, and press the button. The tool will begin moving downward at 10ipm (250mm/min) until it reaches the plate. It will then retract .05" (1mm) and probe again at 1ipm (25mm/min) to get a more accurate reading.

Z zero is then set to the top of the workpiece.

The tool will then retract to the value specified in the Clearance Plane DRO.

Simple auto zero also supports the Material Offset option. See explanation above.

Laser Crosshair Setup

This button sets the spindle 0,0 position to that of a laser crosshair or pointer. Before using, the laser offset distances from the spindle must be set in the M890.m1s macro.
Open the macro in a text editor, and you'll see the first two lines shown below.

```
XOffset = 3.43    'X offset from laser crosshair to spindle
YOffset = 4.322   'Y offset from laser crosshair to spindle
```

Modify the offset values to match the offset distances (from laser to spindle) on your machine. Then save the macro.
When the button is pushed, the DRO's will be set to the offset values, which will place the 0,0 position at the point of the laser crosshair.

Customization

If editing using Screen4, when opening, select Custom option, and use the following size.
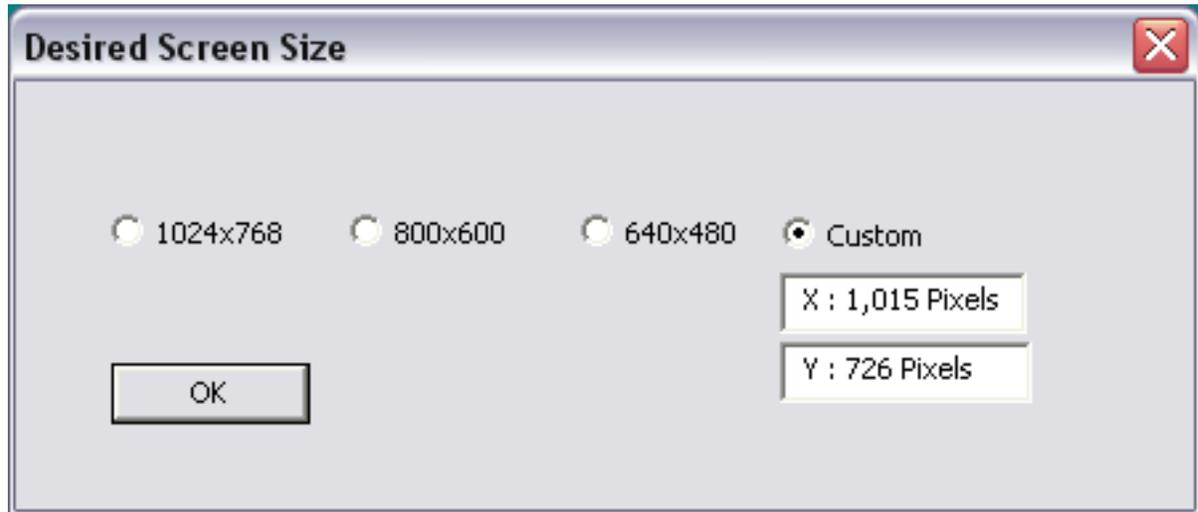
The following User LED's, UserDRO's, and macro's are used in the 2010 screenset.

<u>User DRO's:</u>
1801-1805 - Jog % Increments on Settings tab
1806-1810 - Jog Step Increments on Settings tab
1811 - Fixed Plate X position on Toolchange tab
1812 - Fixed Plate Y position on Toolchange tab
1813 - Calculated Offset for Auto Zero on Toolchange tab (Hidden from user)
1814 - Clearance Plane on Toolchange tab
1815 - Material Offset on Toolchange tab

<u>User LED's:</u>
1854-1859 - G54-G59 Offsets on Offsets tab
1860 - Axis Scaling in Coordinate area
1861 - Machine Coords for Park position on Offsets tab
1862 - SafeZ for Park position om Offsets tab
1863 - Material Offset on Toolchange tab
1865 - Manual Spindle Control on Toolchange tab
1866 – Ignore M6

Macro's:
M6Start.m1s
M6End.m1s
M881.m1s   (Initial Auto Zero)
M882.m1s   (Go To Toolchange Position)
M883.m1s   (Go To Park Position)
M884.m1s   (Ref All)
M885.m1s   (Ref X)
M886.m1s   (Ref Y)
M887.m1s   (Ref Z)
M888.m1s   (Ref A)
M889.m1s   (Simple Auto Zero)
M890.m1s   (Zero Laser Crosshairs)

Extended Diagnostics Screens

At the request of a few users, two screensets are supplied with an "extended" diagnostics screen.
2010x.set, and 2010-4x.set
These screens have additional LED's on them, but they are not tied to any functions. It is up to the user to modify the screensets to utilize these functions.

Referencing (Homing)

The "Ref All Home" button and individual axis Ref buttons call external M codes, and can be modified to suit a particular machine using Operator > VB Script Editor. This allows users to modify their homing procedures, and not have them overwritten if future screen updates become available. The following functions are used to home the axis' in Mach3.

To Home an individual axis, use DoButton(x). Axis numbers are as follows:
X=22
Y=23
Z=24
A=25
B=26
C=27

example:
To Home Z, then X, then Y, the script would be:

DoButton(24)
DoButton(22)
DoButton(23)

To Home multiple axis' at the same time, use RefCombination(x). Add the axis codes together to get the value to use.

X=1
Y=2
Z=4
A=8
B=16
C=32

example:
To home X and Y together, then Z, use:

RefCombination(3)
DoButton(24)

It's possible to have the homing script perform other functions as well. The default Ref All script (M884.m1s) included with the 2010 screenset contains code to move the axis 1" away from the switches, assuming that X and Y home negative, and Z homes positive. These movements are commented out by default. The standard script appears in the VB editor as below.

```
DoButton( 24 )  ' Home Z
'Code "G53 G0 Z-1"
'While IsMoving()
'Wend
DoButton( 23 )  ' Home Y
'Code "G53 G0 Y1"
'While IsMoving()
'Wend
DoButton(22)  ' Home Z
'Code "G53 G0 X1"
'While IsMoving()
'Wend
```

Removing the ' from the start of each line will enable the movements off the switch. The code will then appear as below. Be sure to save the macro when exiting the editor for the changes to take effect.

```
DoButton( 24 )  ' Home Z
Code "G53 G0 Z-1"
While IsMoving()
Wend
DoButton( 23 )  ' Home Y
Code "G53 G0 Y1"
While IsMoving()
Wend
DoButton(22)  ' Home Z
Code "G53 G0 X1"
While IsMoving()
Wend
```

Change Log

v1.0.4 (12/31/10)
Moved macros to external M codes

v1.0.5 (1/12/11)
Added Simple Auto Zero

v1.0.6 (3/4/11)
Added Laser Crosshair Zero
Added toolbar button for Mach3 Mill Addon Wizard
Added 2010 Probing Wizard
Added option to ignore M6
Added Feedrate DRO

v1.0.7 (3/10/11)
Added toolbar button to open G-Code List

v1.0.9 (8/13/11)
Fixed toolbar buttons turning black if clicking off the button and dragging on to it.

V1.1.1 (11/9/11)
Updated Probing Wizard – See Probing Wizard manual
Included 1680x1050 and 1920x1080 versions courtesy of Kyel Smith
Added options for A and B axis MPG jogging on 4 axis screens

V1.1.2 (11/27/11)
Previous release had an incorrect 2010x.set file. Proper file has been included
M881 and M889 macros updated. Fixed a bug where a large Material Offset value could cause the Z axis to crash.

V1.1.3(3/17/12)
French translation and multilingual installer generously donated by Jean-Philippe CIVADE.
M6End and M883 (Park) macros updated to fix bug if Mach3 was in Incremental mode prior to running macros.

V1.1.4(4/13/12)
Added 4 axis versions of 1680x1050 and 1920x1080 Screensets.