Setup procedure for Tiag Micromill Tool length setup

First off let me say a very big thank you to Bernie (beriebro) for his excellent post on the following link
http://www.cnczone.com/forums/showthread.php?t=56079
A big thanks to Hood for helping me with my homing routine
http://www.machsupport.com/forum/index.php/topic,9392.0.html
and to all the others that I may have forgotten

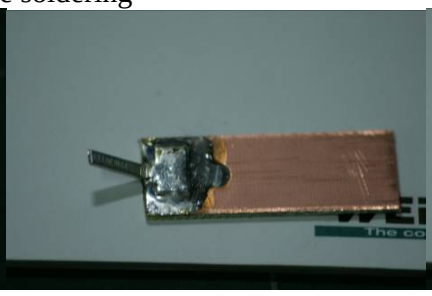
Pretty much every thing I have done is based on Bernie's post,
http://www.cnczone.com/forums/showthread.php?t=56079
so be sure to read and follow the instructions as Bernie and the others users described.

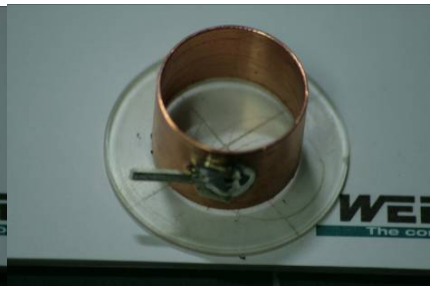
I have deviated from this in a few ways which I will now describe:

NB!! **Warning**
Please do not use these instructions if you are not capable of some basic electronics, as I can't accept any responsibility if it all goes wrong.

1. Connecting the touch probe.
The best way of doing this is to use a BOB (Break Out Board), but if you don't have one, the following will guide you in making a simple connection.
In the file ***TiagMod1v0.pdf*** is the schematic of what I have done so far.
For the touch probe you only need a resistor and a cap, as shown connected to pin 15 of the parallel port (I used LPT2 for this). It is important to note that this is a direct connection to the parallel port so please take care and double check you connections.
As this is part of the documentation for my machine, the Estop and Motor control is also shown.
You can ignore these. (Motor control and Estop)
A common mistake is to connect the crock clip to the input (P15) and the probe to GND, when the croc clip gets connected to the machine the probe shows triggered all the time, because all machines should be grounded, however on the Tiag the spindle holder and the table are anodized and this has good electrical isolation
I used "Faston" or "spade" connectors, to connect the wire to the plate and pipe, on the male, I cut the insulation off and bent it flat before soldering

"Faston" or "spade" connectors


Touch Plate


Copper tube

This makes it easier to change from one to the other. Using this method you can fit plates to the machine for "homing" and then just plug the probe wire onto it when you need to home that axis.
I am not going into much detail here because most of it is covered in Bernie's post

2. Software and screen layouts.
Attached is the screen set ***MachBlueTiagmm.set***, ***MachBlueTiag.set***, ***MachBlue_programRun.jpg***, which have been changed in the following way

a. Added a label for the active co-ordinate system (COSYS)
b. Added 2 buttons for changing between G54 and G55
c. Changed VB code for metric

If you are using inches then use the ***MachBlueTiag.set***
Copy these files into the c:\mach3 directory.(Be sure to load the one you choose from the "View Load Screens" menu in Mach3)

d. The ***MachBlue_programRun.jpg*** is the program screen (1) image that has been changed to allow the display of active co-ordinate system (COSYS) and the G54 and G55 buttons. Copy this file into C:\mach3\Bitmaps\MachTestScreens to overwrite the old one.

If anyone is interested how I did this see ***How to hints for Mach3Screen mods.pdf*** for a rough guide This just a little guide I made to remind myself when the old age sets in, and is by no means comprehensive.

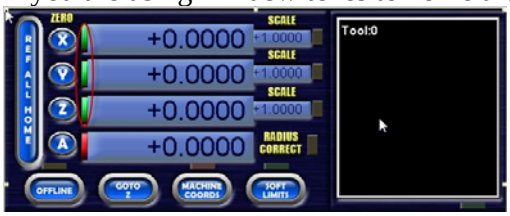
e. Changed the VB script on the "Set tool offset" on the "offset page" to be the same as the Zdown arrow to touch plate.

3. Using the touch probe for tool change
Once all this has been done and you are happy it works. (touch probe works)
I use it as follows:

First some basics.
Machine co-ordinates are stored in G54 (and G53) (*I am still a bit confused how these 2 are linked together*)
First make sure you are in G54

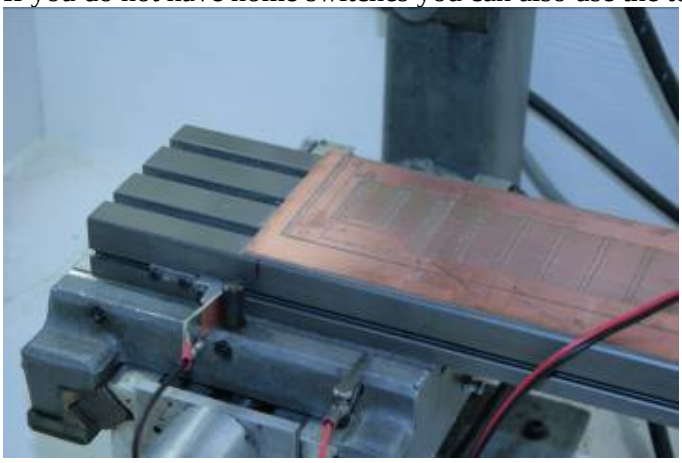
Or you can go look on the offsets page

If you are using limit switches to home the machine, do so



If your homing routine doesn't Ref all home then click on Ref all home (the Homed LED next to the Axis DRO turns green and XYZ must be Zero)

If you do not have home switches you can also use the touch plate to do this



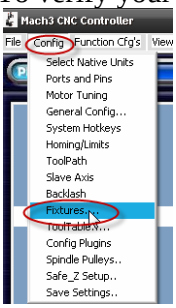
By placing the touch plate between stops and clicking the corresponding button from the touch menu

The X+ in my case, but it may differ on your machine depending on your direction setup.

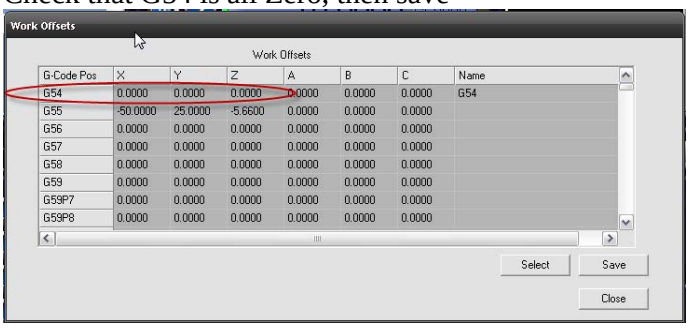
Do this on all axis, then click ref all Home and make sure you are in G54 and that all axis are on zero

The z axis must be zeroed in the upper position (furthest away from the table or job)

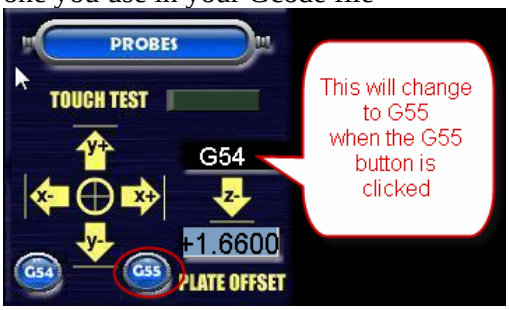
To verify your work offset, click “config” then “fixtures”



Check that G54 is all Zero, then save



Now that the machine is homed you can proceed to set up your work offset for your job
First select work offset G55 or any other one accept G53 and G54 as long as it corresponds with the one you use in your Gcode file

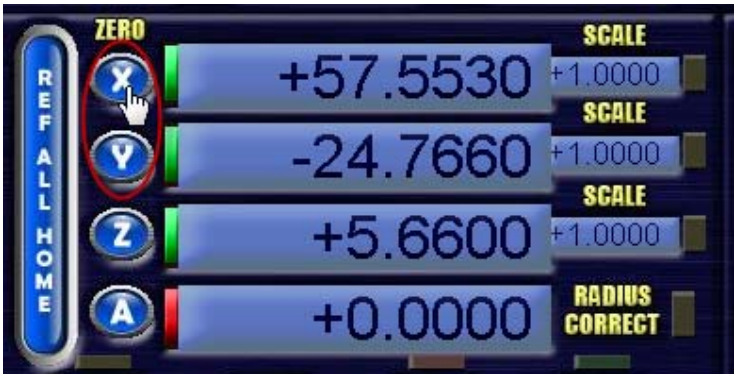


Test your touch system to make sure the Touch LED comes on when the copper tube makes contact with the tool, (the tool used doesn’t matter in this case however be careful with 2 flute cutters as the center may not be correct, I use the back end of a broken PCB drill, or a dowel pin)

then using the Copper tube place the target on your jobs X0 and Y0 intersection (this must correspond with your Gcode file origin.) NB! Hold it firm so it doesn't move

Click the center finder button

The machine will move in 4 directions and return to the center of the tube



Once the machine comes to a stand still, Click the X and Y zero buttons

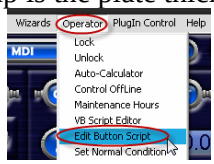
Now change over to the touch plate  Make sure you have the correct 

touch plate height in (You may have a couple of different shaped ones)

Now click the Z down arrow 

or Set tool offset in the "Offsset page" 

Mach3 will zero the z axis and move up the distance it moves up is the plate thickness plus 6.4mm or .25 inch

These values can be changed in the VB script in Mach3 


```
If PlateOffset <> 0 Then
        Call SetDro (2, PlateOffset)    'set the Z axis DRO to  plate thickness
        Code "G4 P0.25"                 'Pause for Dro to update
        ZNew = PlateOffset + 0.25       'plate size plus 1/4 inch
        Code "G0 Z" &ZNew               'put the Z retract height you want here
        Code "(Z axis is now zeroed)"   'puts this message in the status bar
End If
```

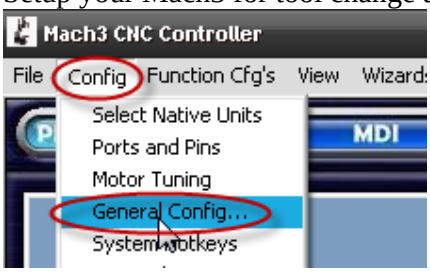
To be safe move home, for this I use The MDI and type M26 (I will add a home button later)

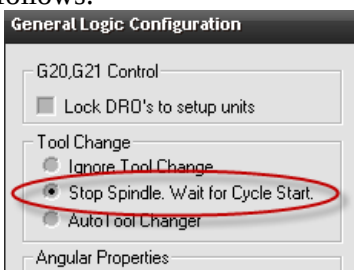
My M26 homing code (Why M26, it is used on some of my other controllers, so I used it)
Attached file **M26.m1s** must be copied into the Mach3\macros\Mach3mill directory
This VB script could be included in M6start macro, but I prefer not to that way I can go else where for tool changes if need be

Code for M26.m1s macro

```
Feed=GetDRO(18)
ofset=GetDRO(46)
code "G53 G0 Z0"
While IsMoving
Sleep (10)
Wend
Code "G53 G0 X0 Y0"
While IsMoving
Sleep (10)
Wend
Call SetDRO(18,Feed)
Code ("ofset")
End
```

Setup your Mach3 for tool change as follows:





Remember to check C:\mach3\macros\Mach3mill directory for the following files or else it will not work

M26.m1s ‘attached to this post
M6Start.m1s ‘standard with Mach3
M6End.m1s ‘standard with Mach3

A Typical program would look and work something like this
(load your Gcode file)

Click cycle start

```
(Program PowerPcbDrill)
(T1 F00 S00 C0.8)
(T2 F00 S00 C1.0)
(T3 F00 S00 C1.5)
(2009/01/03 11:23 PM)
G00 G17 G21 G40 G49 G80 G90 G55   ‘you may want to change some of these, like G20 for inch or G21 for metric
M26 (or Z38.1 (Z1.5 inch))        ‘Homes machine or (moves z up 1.5 inches)
M06 T01 (Drill 0.8)
```

‘The program will stop here

Remember to check for G55

Insert your cutter (tool 1) in the tool holder/collet
Jog Z down close to your jobs Z0 point (possibly X and Y as well)
Place the croc clip on the cutter
Touch the plate on the cutter and make sure the Touch LED comes on (turns green)

Click the Z down arrow 

holding the touch plate firm on the job Zero point

Once the tool has touched you can Click 

The machine will now move to the last position in the program (in this case home)
NB!! You can leave out the M26 (home) command and simply move the Z up enough from the job to enable a tool change. (I do this when the home is far from the job and I want to save time.)

G00 X-41.15 Y-7.62 M03 S10000 (all Sizes are Metric)
G00 Z2
Z0.5
G01 Z-1.8
G00 Z2
.
.
.
M26
M06 T02 (Drill 1.0) *'The same applies as above*
X-19.81 Y20.32 M03 S10000
G00 Z2
Z0.5
G01 Z-1.8
G00 Z2
X-27.43 Y20.32
.
.
.
M26
M06 T03 (Drill 1.5)
X-50.29 Y-7.62 M03 S10000
G00 Z2
Z0.5
G01 Z-1.8
G00 Z2
.
.
.

M26
M30

Good luck
Friedrich