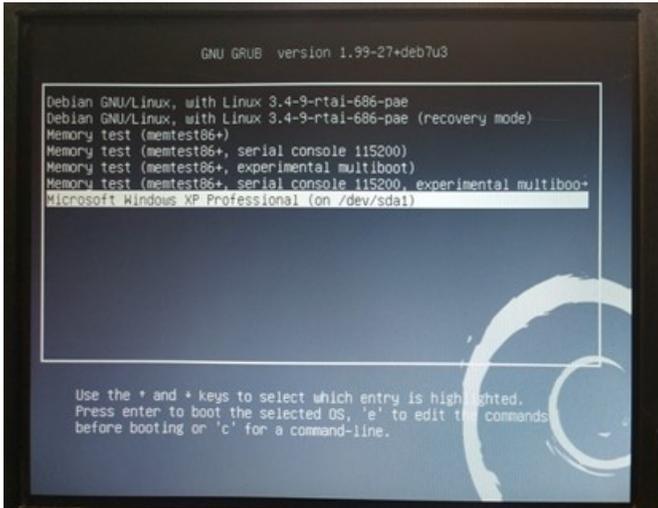


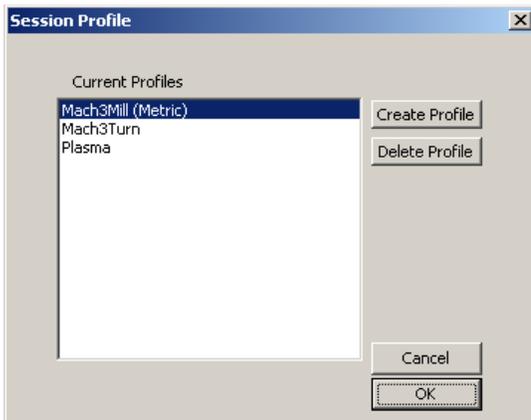
## Protospace Mini CNC User Basic Instructions:

### Step 1 – Starting up the Computer and CNC

1. To run the Mini CNC, boot the PC into XP.



2. The Mach3 application will autostart (Automatically launch).
3. Select Mach3Mill (Metric). Click OK

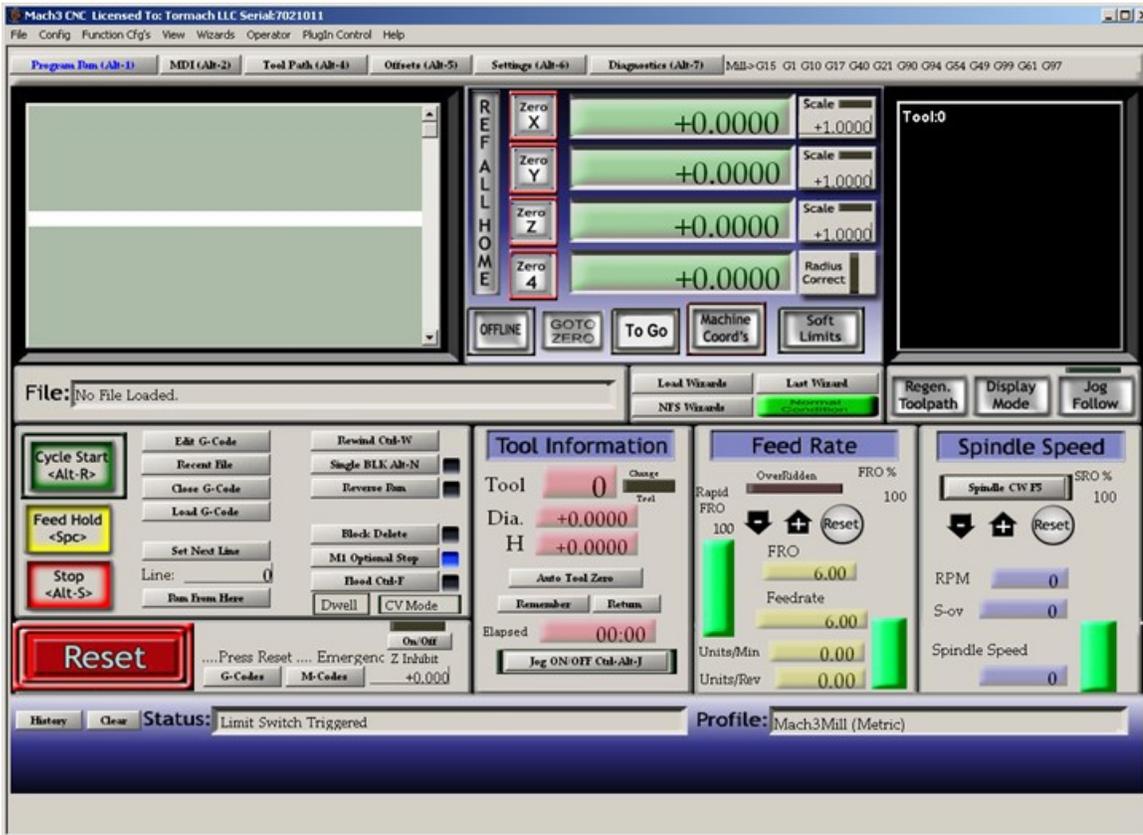


4. Turn on the power on the 3040 and make sure that the E-Switch is not pressed.



## Step 2 – Introduction to Mach 3

You should now see the Initial screen:



### 5. Clear any error messages in Status

The first thing is to Notice Red flashing border on the Reset button and Status message.

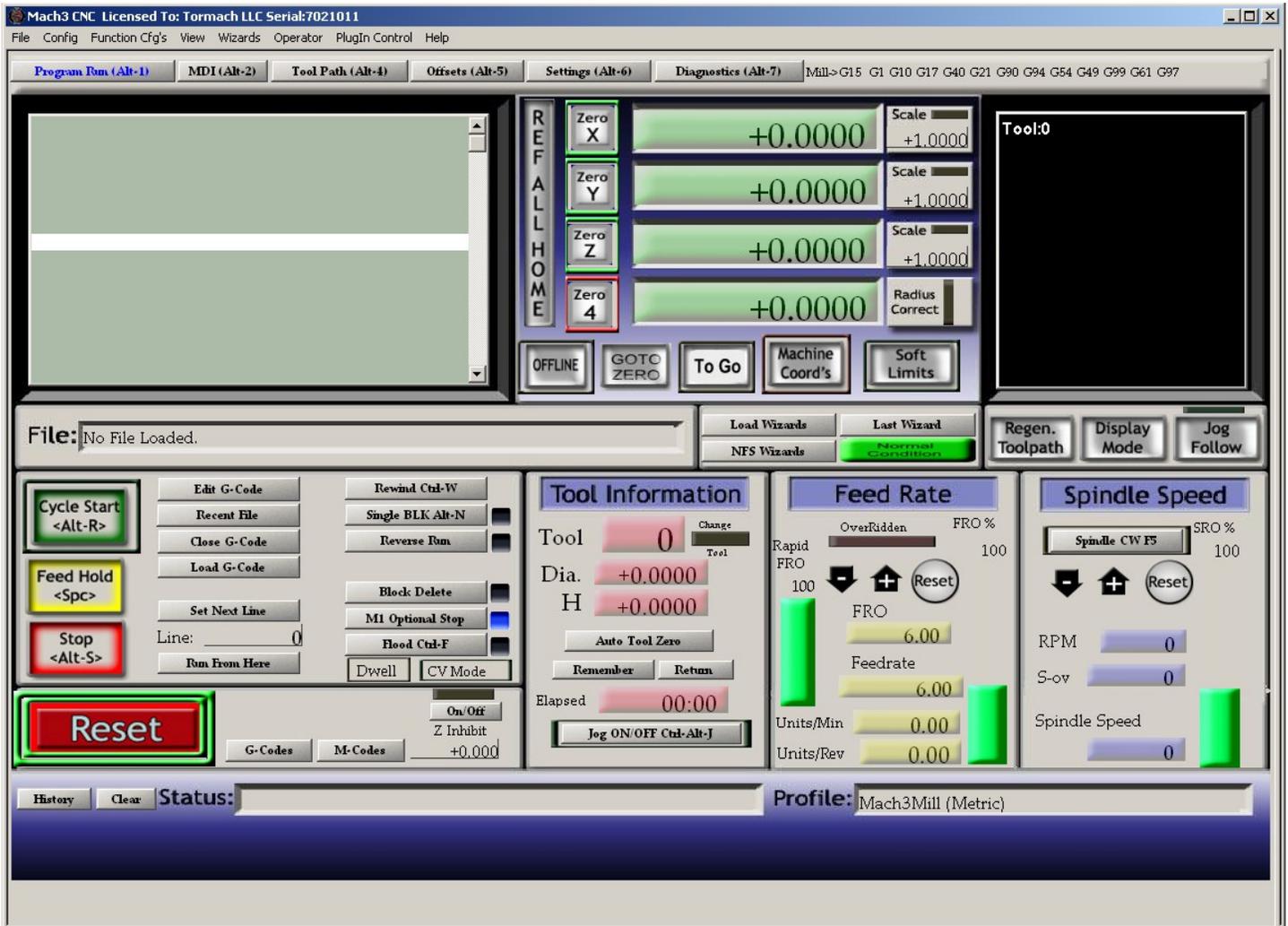
The example shows: “Limit Switch Triggered”. And the scrolling message says “Press Reset... Emergency mode Active. Emergency Z Inhibit”.



You may need to manually spin the knobs on the 3 axis to move the carriages off of the limit switches.

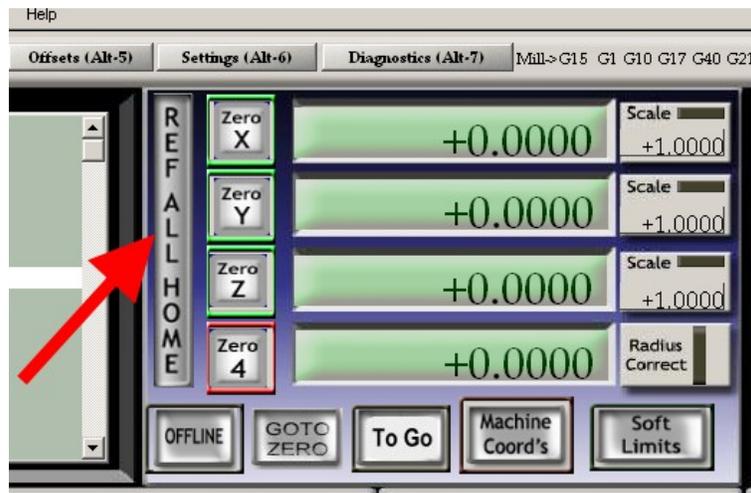
Keep clicking reset, and resolving the issues, until the status is clear of any messages. The Border on the reset button should also be green.



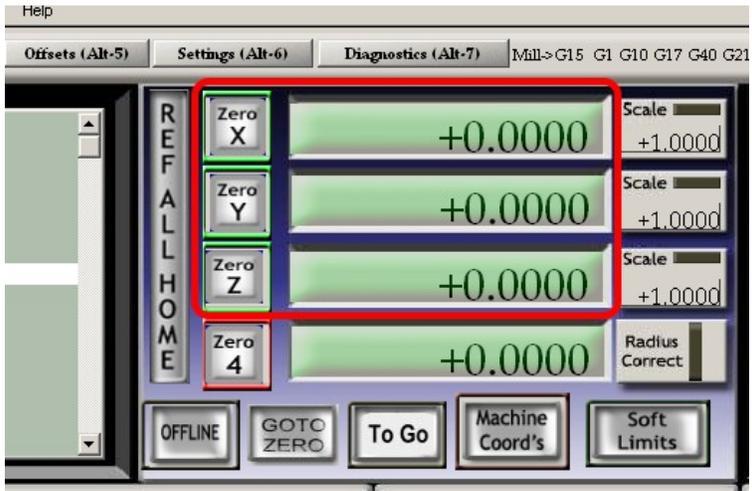


## 6. Home the carriages

Click in the "REF ALL HOME" button.



The 3 carriages will seek their home positions and the 3 buttons will go green and the position will show +0.0000.



### Step 3 - Getting ready to mill your piece

7. Mount your piece on the milling table.

You need to be careful of

8. Move carriage to a new home position

Note that the travel limit switches are now in a “Dead Man” mode. If you accidentally drive the carriage too far and it trips a switch, the CNC will go into error mode and you will need to manually resolve.

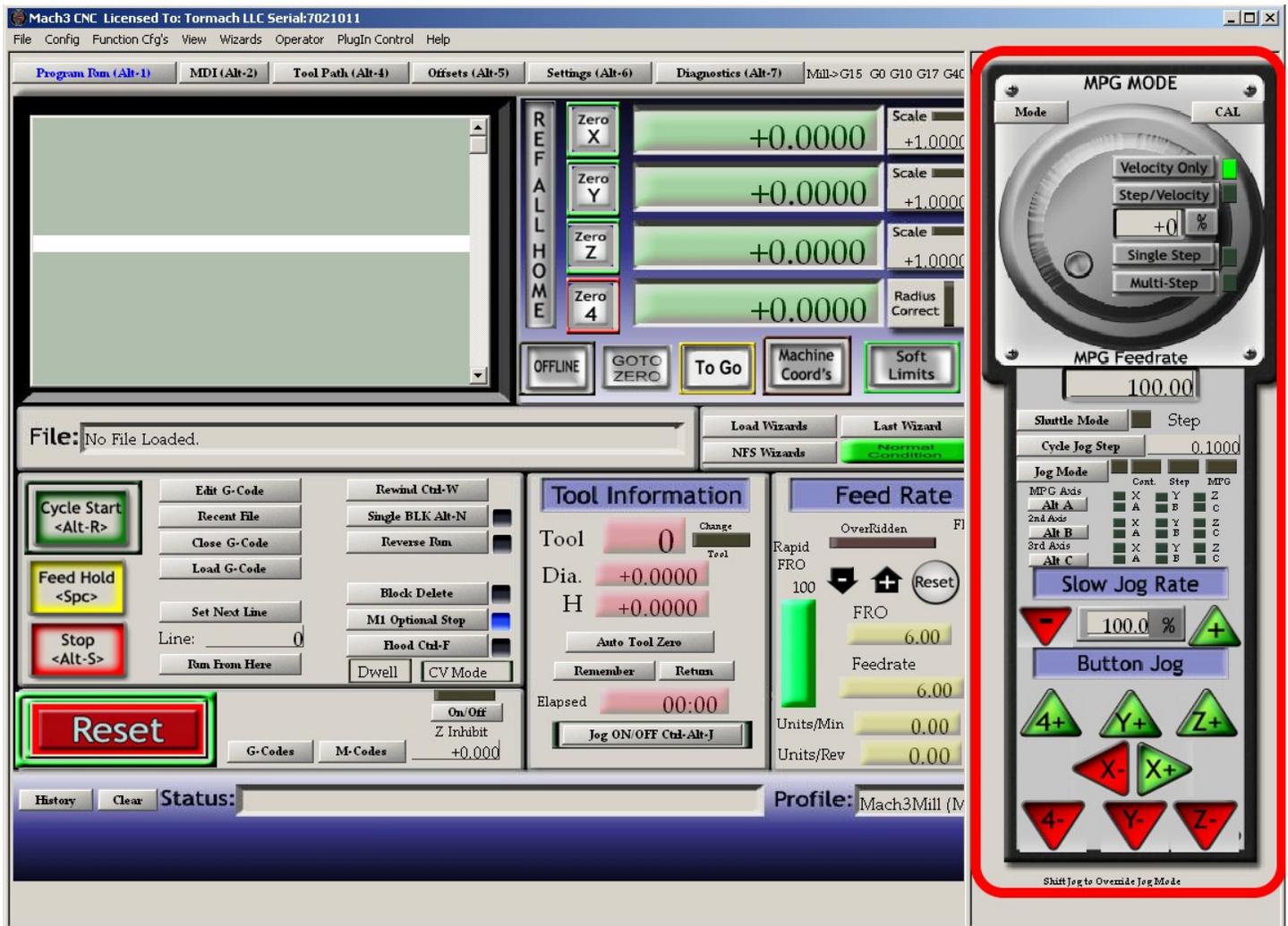
To assist you in preventing this problem, enable the “Soft Limits” button.



The behaviour will now be that the controller will not allow you to go out of limits. You will hear a noticeable slowing down in the motion if you try to drive it too close to the edge.

Now, you can manually move the carriage to a new position, like the a new home position on the corner of your work piece.

The Secret Key => TAB. This will turn on/off the manual controls:



Note that Green is Positive direction and Red is negative.

Also, the direction of the arrows match the direction of the 3 axis (left => left, up => up, etc)

Once you have positioned the carriage in the perfect position, click on each of the "Zero X" buttons.

Mach3 CNC Licensed To: Tormach LLC Serial:7021011

File Config Function Cfg's View Wizards Operator PlugIn Control Help

Program Run (Alt-1) MDI (Alt-2) Tool Path (Alt-4) Offsets (Alt-5) Settings (Alt-6) Diagnostics (Alt-7) Mill->G15 G1 G10 G17 G40 G21 G90 G94 G54 G49 G99 G61 G97

REF  
A  
L  
L  
H  
O  
M  
E

Zero X +0.0000 Scale +1.0000

Zero Y +0.0000 Scale +1.0000

Zero Z +0.0000 Scale +1.0000

Zero 4 +0.0000 Radius Correct

OFFLINE GOTO ZERO To Go Machine Coord's Soft Limits

Tool:0

File: No File Loaded. Load Wizards Last Wizard Regen. Toolpath Display Mode Jog Follow

Cycle Start <Alt-R>

Feed Hold <Spc>

Stop <Alt-S>

**Reset**

Edit G-Code

Recent File

Close G-Code

Load G-Code

Set Next Line

Line: 0

Run From Here

Rewind Ctrl-W

Single BLK Alt-N

Reverse Run

Block Delete

M1 Optional Stop

Flood Ctrl-F

Dwell CV Mode

On/Off

Z Inhibit +0.0000

**Tool Information**

Tool 0 Change Tool

Dia. +0.0000

H +0.0000

Auto Tool Zero

Remember Return

Elapsed 00:00

Jog ON/OFF Ctrl-Alt-J

**Feed Rate**

OverRidden FRO % 100

Rapid FRO 100

Reset

FRO 6.00

Feedrate 6.00

Units/Min 0.00

Units/Rev 0.00

**Spindle Speed**

Spindle CW F5 SRO % 100

Reset

RPM 0

S-ov 0

Spindle Speed 0

History Clear Status: Profile: Mach3Mill (Metric)

## Step 4 – Determining the Bit height

The tool probe height sensor has been set up to allow you to accurately control the height of your bit.

To set the bit height, follow these steps:

9. Insert that bit of your choice in the chuck.
10. Attach the alligator lead to the bit.
11. Place the sensor puck under the bit.
12. Click on the “Auto Tool Zero” button. NOTE: Get ready to hit the E-Stop if things go badly. This can damage bits and such.

The Z axis will slowly drop down until it makes contact with the puck, and then retract a short distance. Note that the distance that it retracts can be adjusted, but the default is 1.27mm.

Once it has found the height, Click the “Zero Z” to reset it.

You will now use this new height as a reference for all further work.

Some experimenting would indicate that the distance fro this measured height and your work surface is now about 27.6mm or so. You can do some experimenting yourself to find the true number.

Note as well: This sensor can also be used for edge sensing. This will be investigated in future.

## Step 5 – Loading and editing your G-Code

13. The next step in milling anything is to load a G-Code file.

Note that the creation of a proper G-Code file is up to you. I have documented a very very simple sample in order to demonstrate the concepts of how to use the tool.

To load a g-code file, simply click on “Load G-Code” button.

Now that you have loaded your G-Code, I would like to run you through what I have learned about the dialogs.

- By clicking on the Dialog box, you can then scroll up and down the file. It appears that you cannot edit on the screen, but you can start selecting some other nifty controls within the file.
- To edit the code, you click on the “Edit G-Code” button. This will open the file in a text editor. When you close the file it will automatically load back into Mach3
- Also note if your click on the “Tool Path (Alt-4)” then you can visualize the cut path on each line of the G-Code file.

The screenshot displays the Mach3 CNC software interface. The main window title is "Mach3 CNC Licensed To: Tormach LLC Serial:7021011". The menu bar includes File, Config, Function Cfg's, View, Wizards, Operator, PlugIn Control, and Help. The toolbar shows various function keys: Program Run (Alt-1), MDI (Alt-2), Tool Path (Alt-4), Offsets (Alt-5), Settings (Alt-6), and Diagnostics (Alt-7). The main display area is divided into several sections:

- G-Code Editor:** A text area showing the loaded G-code program:

```
0:G54
1:g1 x0 v0 z0 f10000
2:g1 z-26 f150
3:G1 x0 v-10 f150
4:G1 x2 v-10
5:G1 x2 v0
6:: Second set
7:G1 x4 v0
8:G1 x4 v-10
9:G1 x6 v-10
10:G1 x6 v0
11:: Third set
12:G1 x8 v0
13:G1 x8 v-10
14:G1 x10 v-10
15:G1 x10 v0
```
- Coordinate Indicators:** A panel showing the current coordinates for X, Y, Z, and the tool number (4). All are set to +0.0000.
- Program Limits:** A panel showing the range for X, Y, Z, and the tool number (4). The ranges are: X Range (+0.0000 to +10.0000), Y Range (-10.0000 to +0.0000), Z Range (-26.0000 to +0.0000), and 4 Range (+0.0000 to +0.0000). The mode is set to "Absolute Coordinates".
- Control Panel:** A panel with buttons for Simulate Program Run, Run From Here, Estimated Program Run Time (0:00:00:00), Cycle Start (Alt-R), Rewind (Ctrl-W), Single BLK (Alt-N), Feed Hold (<Spc>), Stop (Alt-S), and Reverse Run. A large red "Reset" button is also present.
- Tool Path Visualization:** A 3D view showing the tool path for the loaded G-code. The path is visualized as a series of vertical lines, indicating the tool's movement along the X-axis.

The status bar at the bottom shows "History", "Clear", "Status:", and "Profile: Mach3Mill (Metric)".