I wanted to design a digitizing probe for my CNC that could take advantage of the simple digitizer application that is included in Mach3. The concept of a digitizer is very simple. Instead of giving your CNC coordinates and having the machine blindly go to them, you give the machine a grid to feel around, when it 'feels' something, Mach3 records this point and writes it to a file.

To achieve this, I designed a sort of celtic switch that if any of the six points lose contact, the circuit is broken. Once I had the simple design below, everything else was just figuring out how to hold it all together.

I used the same 1/2" OD aluminum tube that I did for the pen holder that fits perfectly into the collet of my router (always being sure to unplug the power). Glue a tube approximately 2.5" long into the top layer of MDF. To hold the circuit together when it is not touching anything, I used a sliding closet door spring as it just happens to fit perfectly into the aluminum tube with very little modification.

For this document, I substitute a piece of 1/2" MDF in place of 8 layers of chip board that really have no consequence on function so that it is easy to CNC cut.

Once all of the components are assembled, you simply bolt together the body, compressing the internal spring. Now you just need to introduce Mach3 to your new digitizer!
These are the patterns for the MDF and 1/16" chip board parts to build the digitizer. These represent the basic geometry at work inside of the digitizer. The CAD versions of these parts will be available at http://www.grunblau.com/downloadsBmo.htm for those who wish to CNC these parts.

**HARDWARE AND SOFTWARE SETUP**

You will hook one of your wires up to Pin # 10 (or any other open pin on your breakout board) and the other wire to the GND.

**INTERFACING WITH MACH3**

When properly setup, you can go to the Diagnostics tab in Mach3 and check to see if your digitizing probe is properly configured. When you touch the tip of the probe, you should see this indicator turn on, when you let go, it should turn off.

Go to > Wizards > Pick Wizard... > Digitizing. This will bring up a dialogue box that will ask you for the following inputs:

- **Width (x) of digitizing area:** +0.0"  < This is the size of the part in the X axis from zero
- **Height (y) of digitizing area:** +0.0"  < This is the size of the part in the Y axis from zero
- **Z axis travel height:** ... the safe travel height for the probe from zero
- **Z axis Probe Depth:** +0.0"  < This is how high to probe down to from zero
- **X Stepover:** +0.0"  < This is the resolution of the points probed in X axis
- **Y Stepover:** +0.0"  < This is the resolution of the points probed in Y axis
- **Feedrate:** +0.0  < This is how fast the machine moves

Make sure that you do not set a feedrate that is too fast. You want your CNC to be able to have time to decelerate once it touches something. After setting the desired parameters, click create and load g-code. Zero your machine. A good first test is to run it in the air and manually touch the probe to make sure everything is working properly. Mach3 will save a text file with all of the points automatically. Depending on what software you are using, there are plugins available for importing the point cloud data.

**PORTS AND PINS**

Navigate to Config > Port & Pins > Inputs tab and pan down on the menu until you see the Probe signal. Check Enable and set the Port to #1 and the pin to 10 (or alternate pin #) and check Active Low. Click OK.