MACHMOTION

Tool Setup

Using MachMotion's CNC Control

12/23/2010

Everything you need to know to setup your tooling in Mach3.

MachMotion.com
http://www.machmotion.com
14518 County Road 7240, Newburg, MO 65550
(573) 368-7399 • Fax (573) 341-2672

Initial Tool Setup

The MachMotion CNC control is designed to be as simple to operate as possible. This manual leads you along in a step by step procedure to setup the tooling on your machine. If you already have existing tooling setup, skip to *Future Tool Setup* to see how to add additional tools.

Note: This manual assumes that you are using the Ultimate Screen provided by MachMotion.

Follow the steps below to setup your tooling:

1. Start the Mach3 software.

2. Reference or home your machine by selecting **Ref Home** on the bottom menu bar. Then press **Ref Home** in the sub menu to home all the axes at once or you can home each axis individually.

Figure 1 Reference Machine

3. Setup your tool change location.

   Go to the **Diagns** screen and then select **Mach Settings**.

Figure 2 Mach Settings Page

Enter in the tool change location for the X, Y, and Z axes. This is based off of machine zero not part zero.

Note: If you do not want an axis to move during a tool change, type 9999 in the location value for that axis.

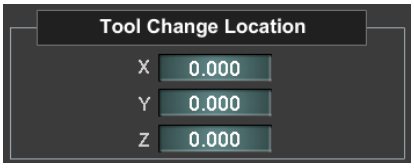

Figure 3 Tool Change Location

4. Return to **Prog Run** screen and press the **Tool Offset** tab.

Figure 4 Tool Offset Tab

Below is a summary of the main features on the **Tool Offset Tab**.

To switch between tools press **T++** or **T—** , then press **Cycle Start**. Notice that the status bar will ask you to press Cycle Start after changing a tool.

Figure 5 Status Bar

Offsets On/Off enables and disables the tool offsets.

Go To Tool Change Position goes to the **machine coordinates** shown in the **Diagns** screen. If 9999 is entered into any of the three fields, that axis will not move when this button is pressed.

Ignore M6 Tool Change disables the M6 macro from running. This allows you to switch between tools without actually moving the machine to the tool change position. For example, assume you are currently in tool 3. To move to tool 5 without moving the machine, select the **Ignore M6 Tool Change** button and then press the **T++** button two times. Your machine is now in the correct tool.

Tool Table button gives an overview of all your tools. You can have a maximum of 253 different tools. In the tool table you can enter in a description for each tool.

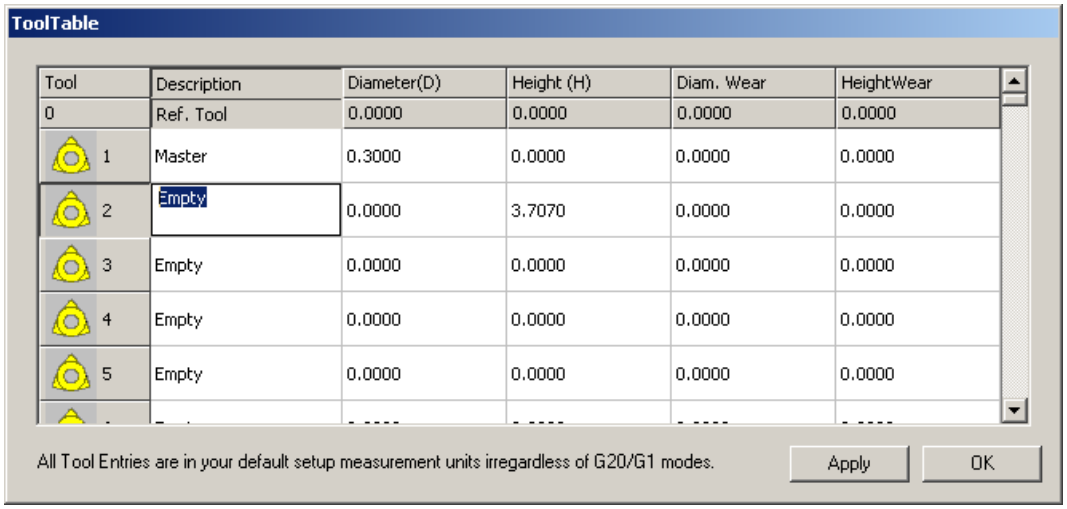

Figure 6 Tool Table

The **Current Gauge Block Height** is only used if you are setting up your machine with a gauge block. Make sure that this value is zero if you aren't using a gauge block.

The basic Tool Offset window can be found on multiple screens. You could setup your machine's tooling in the **Offsets** screen under **Offset Setup**.

Figure 7 Tool Offset Setup under the Offsets Tab

You can also change tools in the **Offsets**, **MDI** and **Diagns** tabs in the small tool change block.

Figure 8 Tool Change Block

5. Increment **T++** to tool 1.

   Press **T++** once and then press **Cycle Start**. The current tool will now be 1.

Figure 9 Tool Setup Window

Use Figure 9 above for a reference for the next few steps.

6. In the field labeled **D**, enter in the tool's diameter.

7. Load your tool into the spindle and then jog the Z axis down to the surface of the part you are zeroing off of. If you are using a gauge block, make sure to enter in the block's height in the gauge block input.

8. With your Z axis at tool zero, select **Offsets** on the bottom menu bar and then zero the Z axis.

Figure 10 Zero the Z Axis

WARNING:

Only do this for the first tool. You will lose your offsets if you zero the Z axis on additional tools.

This is basically your master tool or your base offset. You should not have to zero the Z axis again while setting up tools. The first tool is now setup correctly. Continue to going through the next steps to setup more tools.

9. Press the **T++**. As long as the **Ignore M6 Tool Change** button is not enabled, your machine will move to the tool change location and then increment the tool number. Make sure to press **Cycle Start** after you click on **T++**.

10. Again enter in your tool's diameter in the field labeled **D** (Figure 9).

11. Load your next tool into your spindle and jog the Z axis down to the surface of the part you are zeroing off of.

12. Now press **Set Tool Length**. The correct height will be saved with that tool.

Figure 11 Set Tool Length

Do NOT press Z zero again!

Now your next tool is completely setup.

13. Repeat steps 9 through 12 until all your tools are setup.

Future Tool Setup

To setup additional tools, follow the steps outlined below:

1. Load in a previously setup tool into your spindle. Next use the **T++** or the **T—** buttons to select the tool number of the tool in the spindle.

2. Jog the Z axis down to the surface of the part you are zeroing off of.

3. Zero the work offset by clicking **Offsets**, then **Zero Z** on the bottom menu bar.

WARNING:

DO NOT press Set Tool Length. If you press set tool length you must setup that tool again.

Figure 12 Zero Z

Now you are ready to setup additional tools.

4. To add additional tools, repeat steps 9 through 12 as shown on page 7.