

Manual Automatic Tool Change

Setting Up the Tools

The statement “Manual Automatic Tool Change” sounds like an oxymoronic statement unto itself. Basically what it’s a way to convince Mach 2 that you have an automatic tool changer, when in fact you do not (not yet at least). The auto-change routine includes a pause and a prompt to ask for the proper tool. Somewhere in there it applies the proper tool offset and then resumes the program. When we though this up it cut the production time of our brackets from 3 hours down to 1:20 flat. Now that’s a manufacturing improvement, more so it allows an unskilled operator to “baby sit” the mill without a bunch of training.

This “Manual Automatic Tool Change” series is broken into 3 sections:

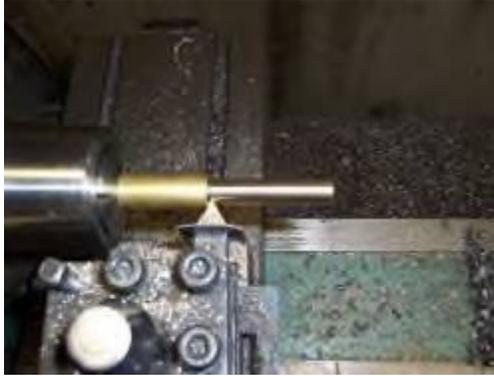
- Prepping the tools (this section)
- Setting up the tool table
- Setting up everything else and making it work

Prepping the tools.

Anytime we talk about swapping tools we need an accurate way to get them in there the same way each time. Basically this is a way to get the drill bits the same, and maybe a few other things that might be helpful.



Start with a piece of solid stock, I choose brass because it solders easily, anything that will solder will work.



Rough turn it down to the diameter + 0.010 of collet you're going to use.



Pilot hole. A good pilot bit is self centering, so it makes things easier. It's worth the 50 cents to pick one up.



Using the same bit that you're going to mount into it, bore a hole about "yo" deep. At least to the depth of the flutes.



That's what "yo" deep looks like.



Part the piece off.



Remember the vice that came with the mill, this is a perfect job for it.



Get it hot, but not red hot, enough to melt solder and maybe a pinch more.



Fill'er up with solder.



I've used both silver and regular solder successfully.



Stick the bit in the solder and press all the way to the bottom twisting all the way. Make sure it is clean and fluxed first. Tinning the bit is optional but couldn't hurt.



When you press down the solder will more than likely SQUIRT OUT. Use caution.



Let it cool and chuck it up. BIT ONLY.

True up the first pinch of the brass, enough to get a better grip. Take your time and make VERY small cuts, mostly with a bit this small.



Chucked up a little better.



Trued.



Test it on the actual R8 collet.



Flip it around and clean up the front.



The bit should run true and straight at this point.



Get a few shaft collars. These are pretty cheap, about 75 cents if bought in bulk.



Shaft collar mounted. Isn't she pretty?...Ok it's a drill bit with a 3/8 brass shank.



More of my collection. Since I use double ended mills I paint the NOT used end with finger nail polish. When I flip the cutter I paint the other end, milling wears the polish off in just a few minutes of operation and it's easy to determine which end to use.

Final Thoughts:

The drill bit used in the pictures is a pinch smaller than 1/8" small bits are always tough to do, larger bits are quite a bit easier and you will not need to cut the little bit to get a better grip. I've done bits as small as 1/16. (Not bragging, just saying you can do it too)

I use a C5 Collet closer, it's actual not necessary, a 3 jaw will work fine.

I chop off the larger diameter bits to shorten them up, then I do them.

I have some popular sizes that are done in both a 3/8" and a 1/2" shank this allows me to speed up production by not having to swap the collet.

Black oxide coated bits don't work, you'll need to remove the bottom section of coating at least.

If a bit snaps while in operation, check the solder joint by heating it up and removing the bit. Check for solder adhesion, you might need to improve your soldering technique.

Setting Up the Tool Table

The statement "Manual Automatic Tool Change" sounds like an oxymoronic statement unto itself. Basically what it's a way to convince Mach 2 that you have an automatic tool changer, when in fact you do not (not yet at least). The auto-change routine includes a pause and a prompt to ask for the proper tool. Somewhere in there it applies the proper tool offset and then resumes the program. When we though this up it cut the production time of our brackets from 3 hours down to 1:20 flat. Now that's a manufacturing improvement, more so it allows an unskilled operator to "baby sit" the mill without a bunch of training.

This "Manual Automatic Tool Change" series is broken into 3 sections:

- Prepping the tools

- Setting up the tool table (This section)
- Setting up everything else and making it work

Setting up the Tool Table

Before we begin with the pictures we need to get some terminology down.

Reference Line: Is an imaginary line that runs parallel with the table just under the quill. It is your Z reading when no offsets are applied; in my case it's 13.75" (Gotta' love the Full Head Z).

Tool Height Offset: A positive number that represents the height of a tool. This number is subtracted from the Reference Line height to get an actual working height.

For example:

When I execute a G00 Z0 with no cutter in the spindle and no offset applied it drops the nose of the collet right to the table. A less technical way of saying "The reference Line is at Z 0".

- Z DRO Start Position: Z 13.75
- Z DRO End Position: Z 0
- Applied Height Offset: 0.00

If there is a cutter in there kiss it good-bye, as it will be planted through the table.

Now, when I mount a cutter say 2" long into the collet, it extends PAST the reference line by 2". When the Tool Height Offset is applied it SUBTRACTS the height of the tool from the Reference Line Height and puts the resulting value into the DRO (numbers on the screen). Now when I execute a G00 Z0 with the offsets APPLIED it will drop the very tip of the cutter to the table.

- Z DRO Start Position WITH offset: Z 11.75 (The original Z 13.75 - Tool Height 2.0)
- Z DRO End Position WITH offset: Z 0
- Applied Height Offset: 2.00

As far as the Mach 2 DRO reads it moved from Z 11.75 to Z 0.

That's it; basically you're telling Mach how long a tool is, nothing magic. The reason it's a table is so you can keep a bunch of tools on file. Mach 2 allows for 255 tools, I use 20 or so, and of those only 10 on a regular basis. But, oh how those 10 make a difference.



All we need to set the height of a tool.

1/2 R-8 Collet, 1/2 double sided end mill, 0.125 precision ground flat stock.



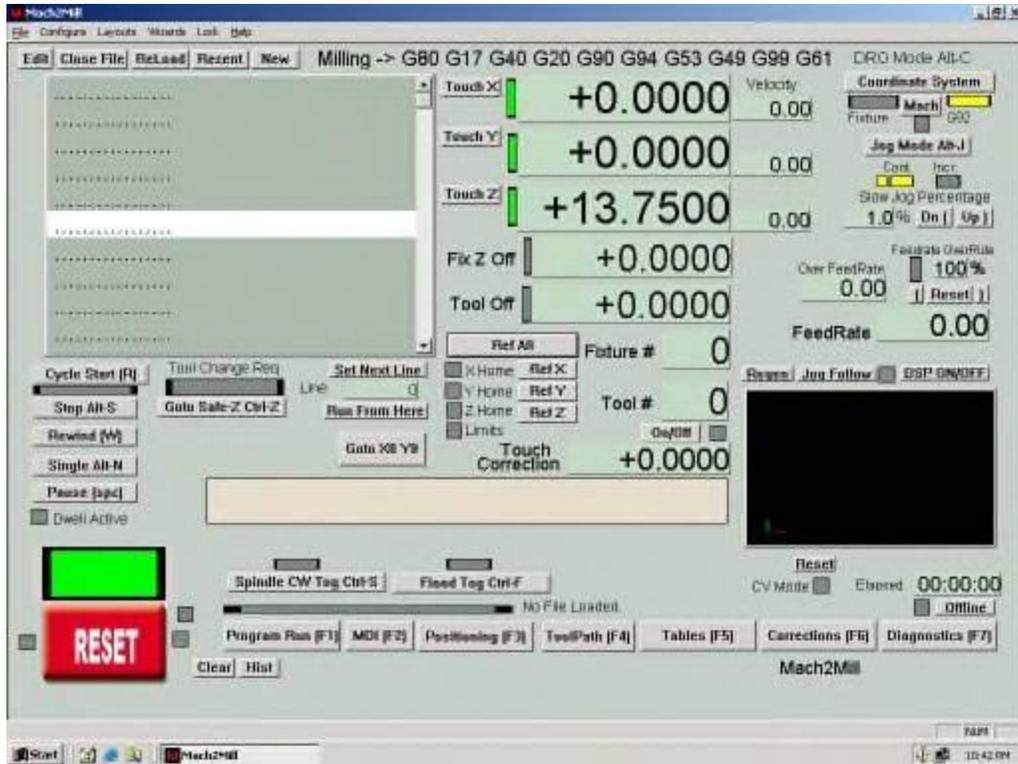
We use the 0.125 P-Ground flat as a shim, you're welcome to use whatever you want. We actually pick this up in 36" lengths and cut them into usable lengths, they make great parallels too.



Start by **Referencing Your Machine and LOCK the QUILL.**

VERY IMPORTANT STEPS

Mount the cutter into the collet, ensuring it makes solid contact with the collet.



This is what the Mach 2 numbers will look like of my machine when "Refed" . Z 13.75 is the important one.



I did a fair amount of customization to the Program Run screen to tune it for the way we work, if you would like the screen design just let me know and I will send it to you. Your screen will look quite a bit different, but the important numbers are the same.



Rapid the cutter down to the table, just above the 0.125 shim



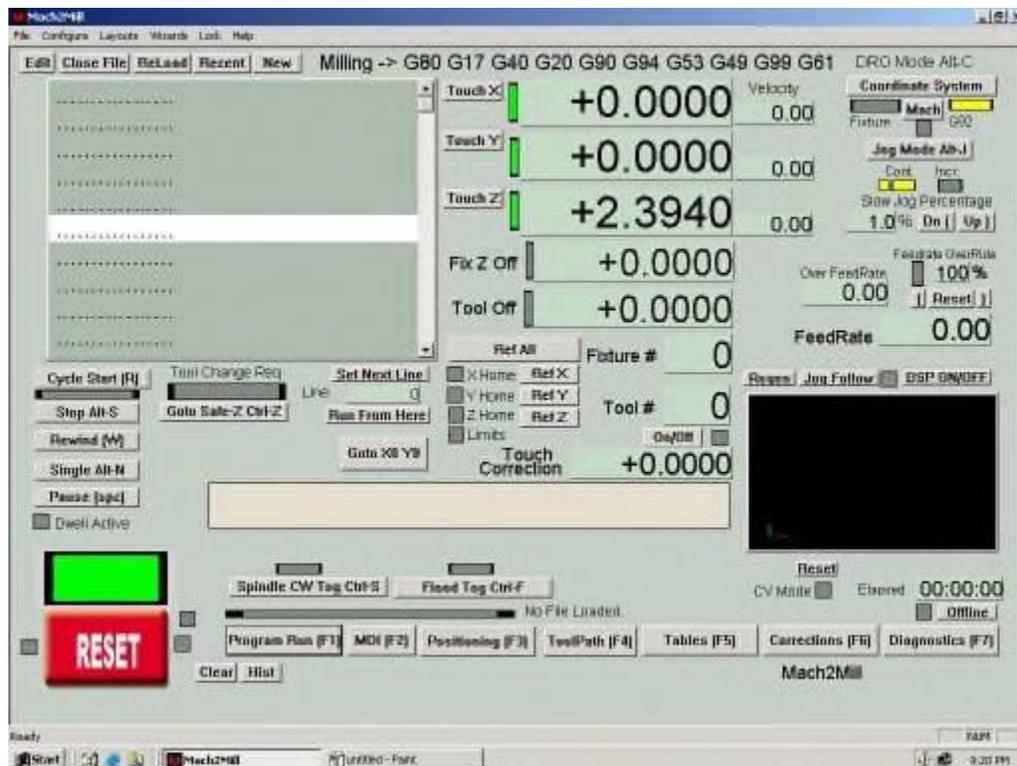
Lift one end of the shim and slowly (less than 1 IPM) lower the cutter the rest of the way, this way it will push it down to the table.



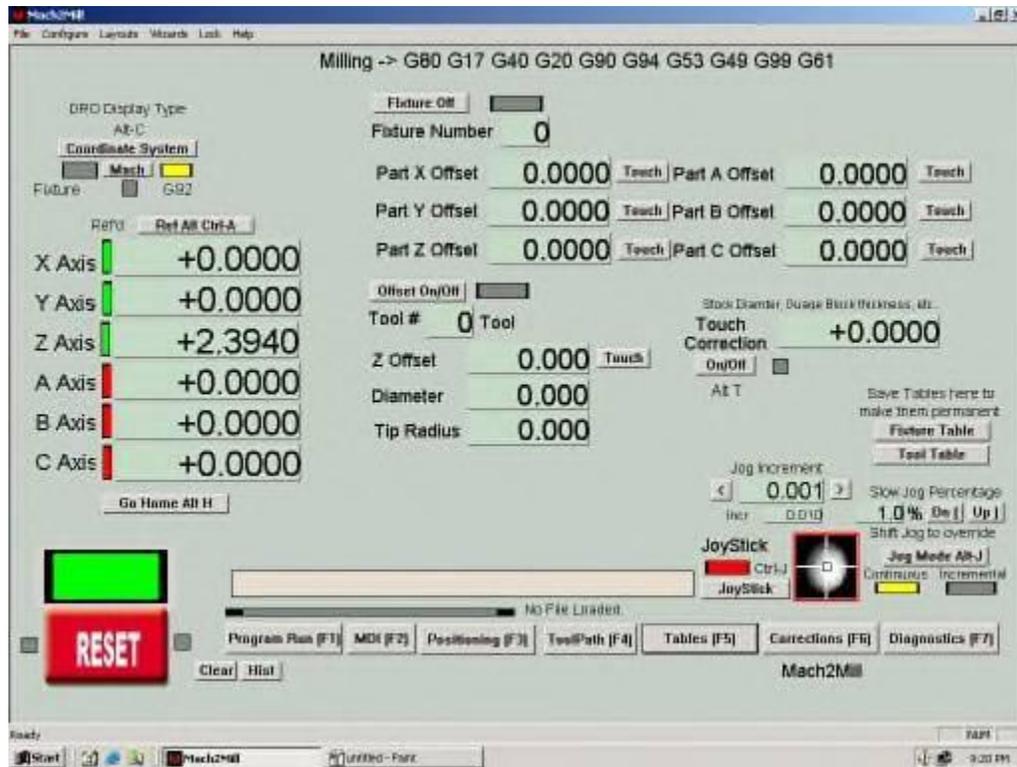
Don't get your finger pinched.



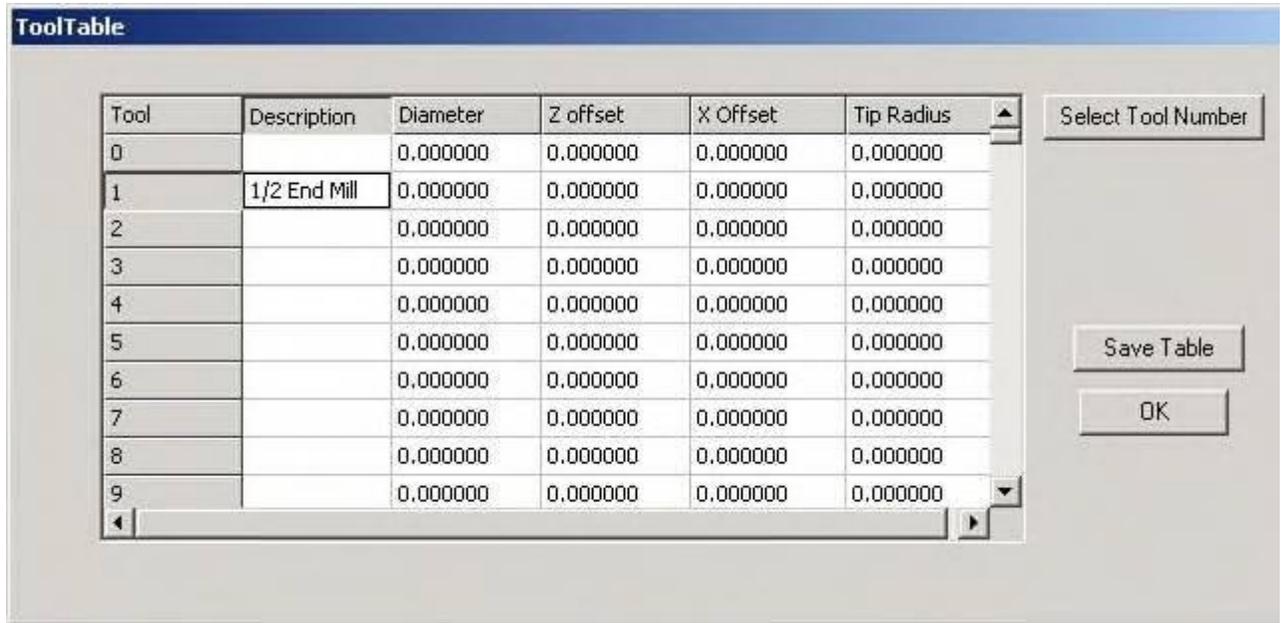
Gently set the cutter on the shim, actually I can slide the shim out, but I feel the cutter drag across the top.



With the end mill sitting on top of a 0.125 shim here are the Mach 2 numbers. Once again focus on the Z Axis.



Change to the Tables screen (F5), it should look more or less like this.

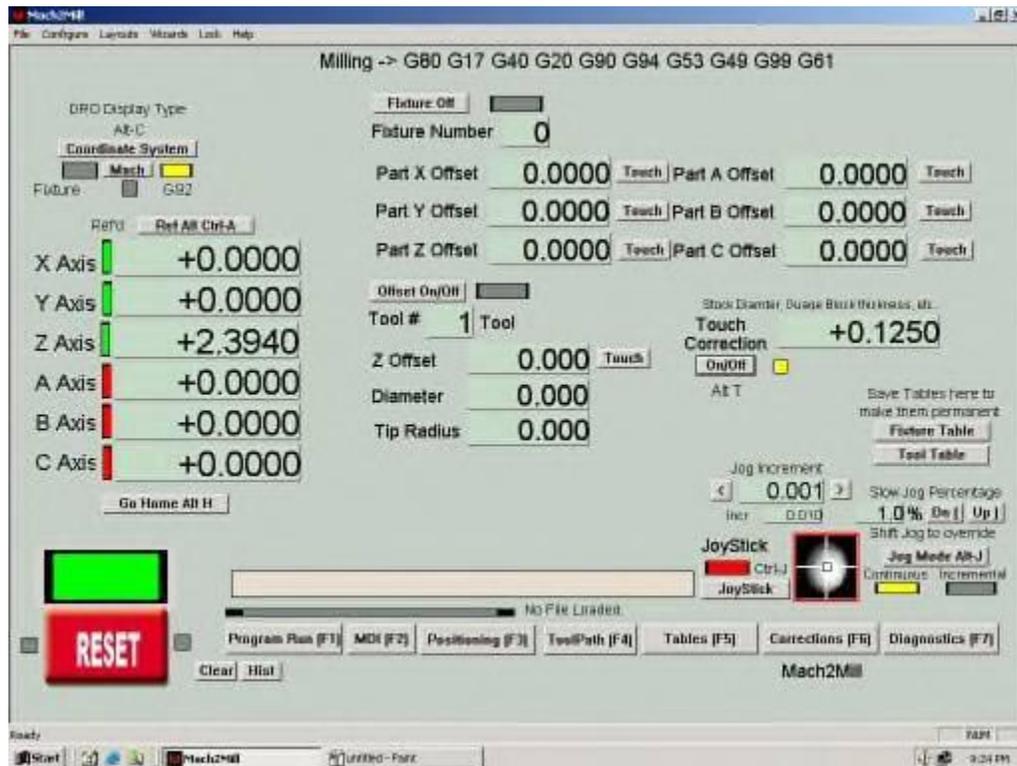


Click on the Tool Table button on the right side of the screen. Enter the name of the tool (whatever you want) and press **Save Table**. Then press **Select Tool Number** and **OK**



Click in the Touch Correction box and enter the thickness of your shim (0.1250 for us).

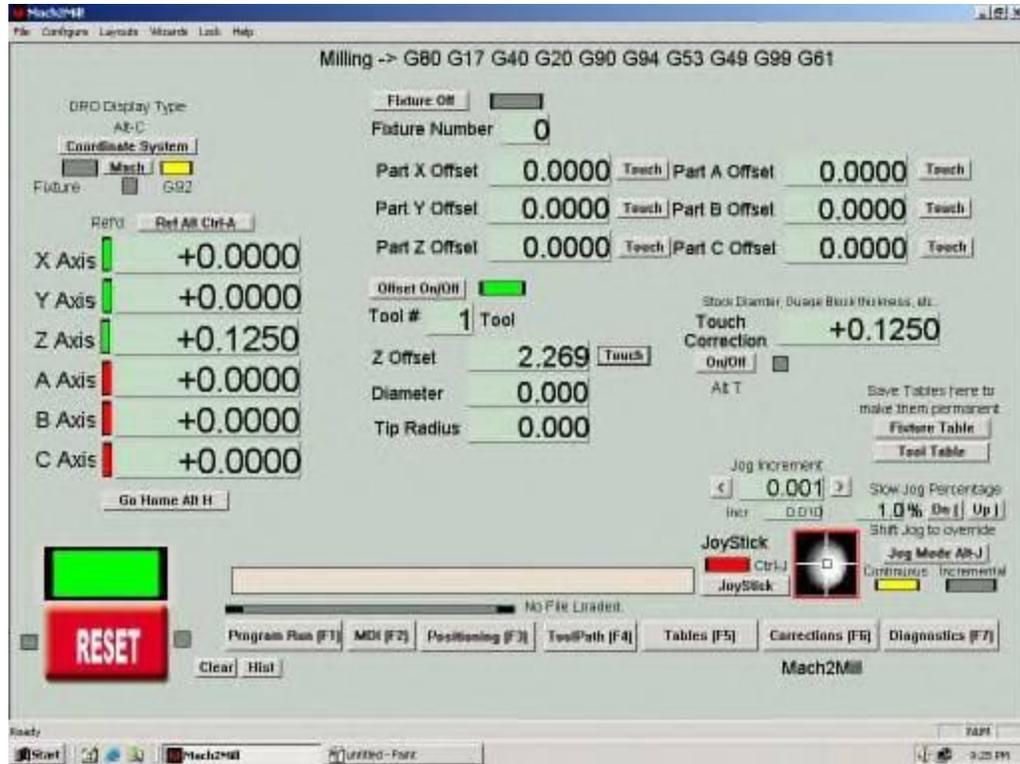
Also notice the tool number has changed to #1



Turn the Touch Correction ON, it will flash yellow.



This is the part people forget.



Click on the **Touch** button in the middle of the screen. that will do the following:

- Calculate the current offset- touch correction
- Apply the offset
- Reset the DRO

Just because I'm superstitious I open the tool table and click **Save Table**, I really do this because the value is cached as a parameter and not saved to disk until you press the **Save Table** button. Under certain conditions, (I've found them) the program will revert to the SAVED value from the disk, so to avoid any problems I save the table after **EACH TOOL** is added/updated.



Our touch correction is **STILL** on, we just caught it in between flashes.

Tool	Description	Diameter	Z offset	X Offset	Tip Radius
0		0.000000	0.000000	0.000000	0.000000
1	1/2 End Mill	0.000000	2.269000	0.000000	0.000000
2		0.000000	0.000000	0.000000	0.000000
3		0.000000	0.000000	0.000000	0.000000
4		0.000000	0.000000	0.000000	0.000000
5		0.000000	0.000000	0.000000	0.000000
6		0.000000	0.000000	0.000000	0.000000
7		0.000000	0.000000	0.000000	0.000000
8		0.000000	0.000000	0.000000	0.000000
9		0.000000	0.000000	0.000000	0.000000

The updated table.



Back to the Program Run screen (F1), we can see the following:

- Tool # 1 is selected

- Tool Offset is 2.2690" and is applied (green led)
- Current position of the cutter is 0.125 (right where we left it on top of the 0.125 shim)
- Touch Correction is off (the flashing light is annoying)

The screenshot shows the 'ToolTable' window with a table of tool parameters. The table has columns for Tool, Description, Diameter, Z offset, X Offset, and Tip Radius. The data is as follows:

Tool	Description	Diameter	Z offset	X Offset	Tip Radius
10	3/8 SPOTTER	0.000000	1.689700	0.000000	0.000000
11	1/4 END MILL	0.000000	1.955400	0.000000	0.000000
12	1/2 END MILL	0.000000	2.308100	0.000000	0.000000
13	5/16 PILOT BIT	0.000000	2.011300	0.000000	0.000000
14	5/16 BIT	0.000000	3.083900	0.000000	0.000000
15	1/2 WOODRUFF	0.000000	2.046100	0.000000	0.000000
16	#6 PILOT	0.000000	5.649800	0.000000	0.000000
17	5/16 Pilot 1/2 Shank	0.000000	1.615900	0.000000	0.000000
18	3/4 End Mill	0.000000	5.730700	0.000000	0.000000
19	7/8 End Mill	0.000000	6.684800	0.000000	0.000000

Control buttons on the right include 'Select Tool Number', 'Save Table', and 'OK'.

We actually use our Tool Table starting at #10, we leave 0-9 for just quick jobs. For tools 0-9 we actually cleared out all the values before we started taking screen shots.

If you look over our tool table you'll notice one value stands out as being odd. The #6 pilot has an offset of 5.6" which is pretty darn long for a little tiny bit. Well, in the last section you'll remember that we "shanked" a pretty small bit, i.e. a new #6 pilot, the old one was lost in a tragic milling accident and we needed to use the chuck to complete the run. We simply stopped the program, chucked up the bit, reset the table and did a "Run From Here" start.

Conclusion:

At this point just repeat the procedure for each tool, it's actually goes pretty quickly. About 3-4 minutes per tool if you take you time.

Final Thoughts:

Although we didn't go over it, repeatability in the collet is half the battle, applying consistent torque to the draw bar is the key to success along with using the same collet for a tool after it's been setup in the table.

We always set tool height at X0 Y0, we do this for consistency.

When we setup for a production run we do the following:

- Pull all the tools and collets needed for the run.
- Inspect and replace tools or collets as necessary.
- Reset the height of all the tools for the run in the Tool Table, reference the Z Axis between each tool

This little procedure takes us about 30 minutes for our most complex part, we'll then run about 18 hours straight with out a single issue. If for some reason we break the run into 2 or more days and we shut down the machine we repeat the entire procedure again. It's not that we don't trust the repeatability of the referenced machine, it's just a good practice to get into, not to mention good insurance against an avoidable mistake. I know it seems like a pain in the ... but to get consistent results you need to have a consistent methodology.

Making it all work

The statement “Manual Automatic Tool Change” sounds like an oxymoronic statement unto itself. Basically what it’s a way to convince Mach 2 that you have an automatic tool changer, when in fact you do not (not yet at least). The auto-change routine includes a pause and a prompt to ask for the proper tool. Somewhere in there it applies the proper tool offset and then resumes the program. When we though this up it cut the production time of our brackets from 3 hours down to 1:20 flat. Now that’s a manufacturing improvement, more so it allows an unskilled operator to “baby sit” the mill without a bunch of training.

This “Manual Automatic Tool Change” series is broken into 3 sections:

- Prepping the tools
- Setting up the tool table
- Setting up everything else and making it work (This section)

Making it all work

This is the third and final (maybe) section on the “Manual Automatic Tool Change”. In this section we wrap it all up and put everything into play.

Setting up Mach 2 for Automatic Tool Change

Logic Configuration

G20,G21 Control
 Lock DRO's to setup units

Z - Inhibit
 Z - Inhibit On
 Max Depth Units

Tool Change
 Ignore Tool Change
 Stop Spindle. Wait for Cycle Start.
 AutoTool Changer

PWMBase Freq.

Laser Mode. freq by Feedrate %
 Use Spindle Feedback for Feeds
 Persistent DROs
 Disable Gouge/Concavity Checks

Input
 Debouce Interval:
 Index Debounce

Angular Properties
 A-Axis is Angular
 B-Axis is Angular
 C-Axis is Angular
 Unchecked for Linear

Pgm End or M30 or Rewind
 Turn off all outputs
 E-Stop the system
 Perform G92.1
 Remove Origin Offsets
 Remove Tool Offset
 Radius Comp Off

Plasma Mode
 No Angular Discrimination
 FeedOverRide Persist
 Unchecked resets on Stop File
 Persistant Z-Inhibit
 Unchecked resets on rewind

Program Safety
 Program Safety Lockout
 This disables program translation while the External Activation #1 input is activated.

PWM Pulley Control

Current Pulley Set	Max Speed	Minimum Speed
<input type="radio"/> Pulley Ratio #1	<input type="text" value="1000"/>	<input type="text" value="5"/> %
<input type="radio"/> Pulley Ratio #2	<input type="text" value="2000"/>	
<input type="radio"/> Pulley Ratio #3	<input type="text" value="3000"/>	
<input checked="" type="radio"/> Pulley Ratio #4	<input type="text" value="25000"/>	

RPM

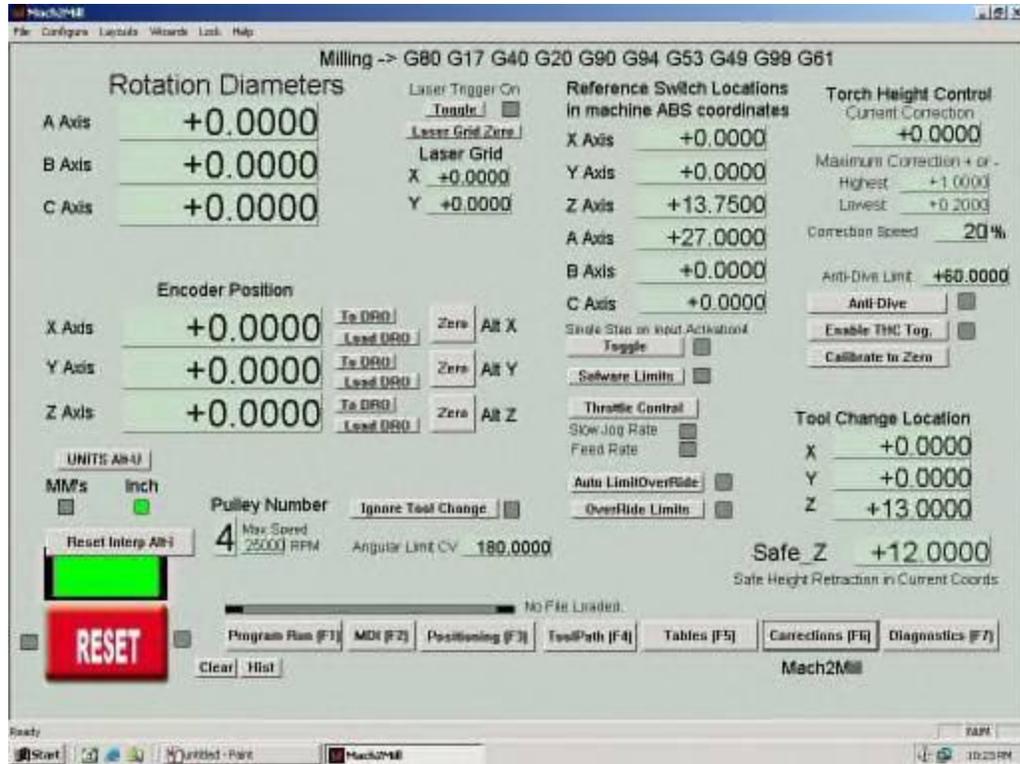
G04 Dwell param in MilliSeconds instead of Seconds

Editor

Serial Output
 ComPort # BaudRate

M01 Control
 Stop on M1 Command

In Mach 2 open up the Logic panel and select "**AutoTool Changer**", click **OK** to save and close the box



Set the **Safe_Z** value to something you like, I use 12.00"

Don't worry about the "Tool change location stuff" those number can all be 0.00

Setting Up The Macros

Find your Mach 2 macros (a Macro is just a small program that runs within another program), the default location is C:\Mach2\macros\Mach2Mill. You can always go through Mach to get them, but I'm a Explorer sort of guy.

You'll need to adjust a couple macros, but before you do that you need to do a few things:

1. Make a folder called Originals in the Mach2Mill folder.
2. Make a **COPY** of **ALL** the macros and put them in the Originals folder

Lets get to work, go back to the Mach2Mill folder and do the following:

Open up **M6Start.m1s** and replace whatever is in there with the following lines.

```
SafeZ = GetSafeZ() ' Load Safe Z Value
Code "G53 G00 Z" & SafeZ 'Goto the safe Z location
Code "M99999" ' Call Macro M99999
```

Open up **M6End.m1s** and delete everything that is in there (if anything).

Create a new file called **M99999.m1s** and place the following lines in it.

```
tool = GetSelectedTool() 'Load the tool number  
SetCurrentTool( tool ) 'Set the tool  
Code "G43 H" & tool 'Set the height
```

Complete = Question ("Change to tool number" & tool)



Make sure you do the following:

- Hit ENTER a few times after the last line of the macro
- Make sure you don't end up saving the macros with a .TXT extension
- The proper extension is .m1s or pronounced **Mach Numeral ONE** Script

Testing It



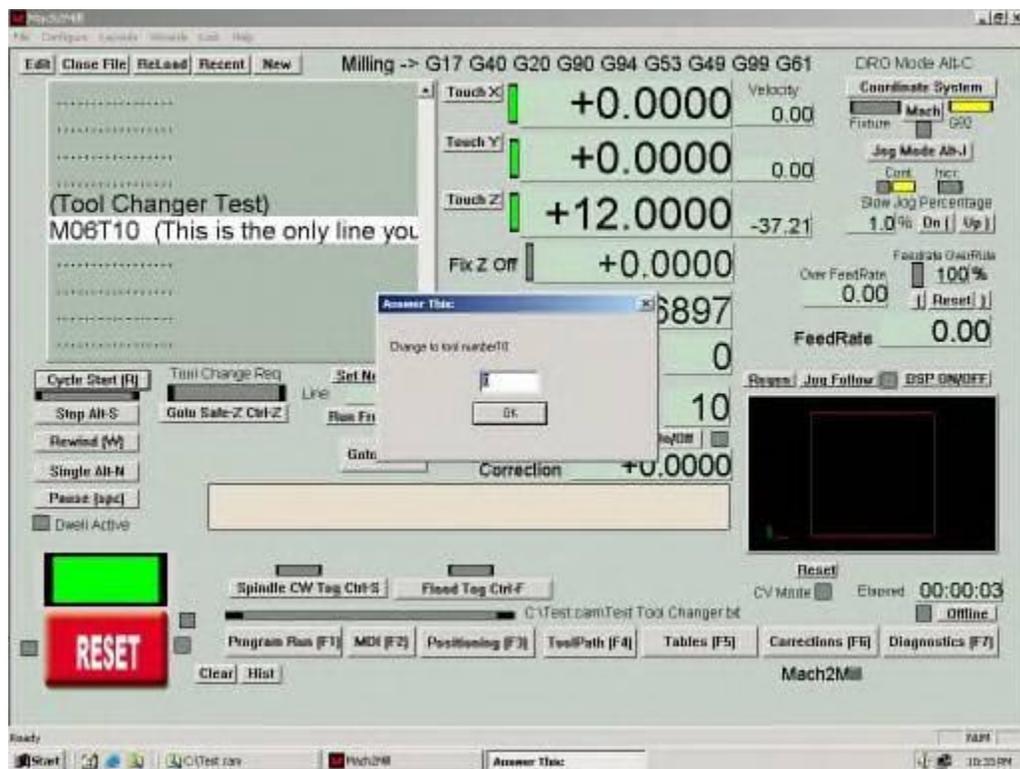
Make a small program with the following line in it:

M06T<Programmed Tool Number>

Load the program

Things to notice:

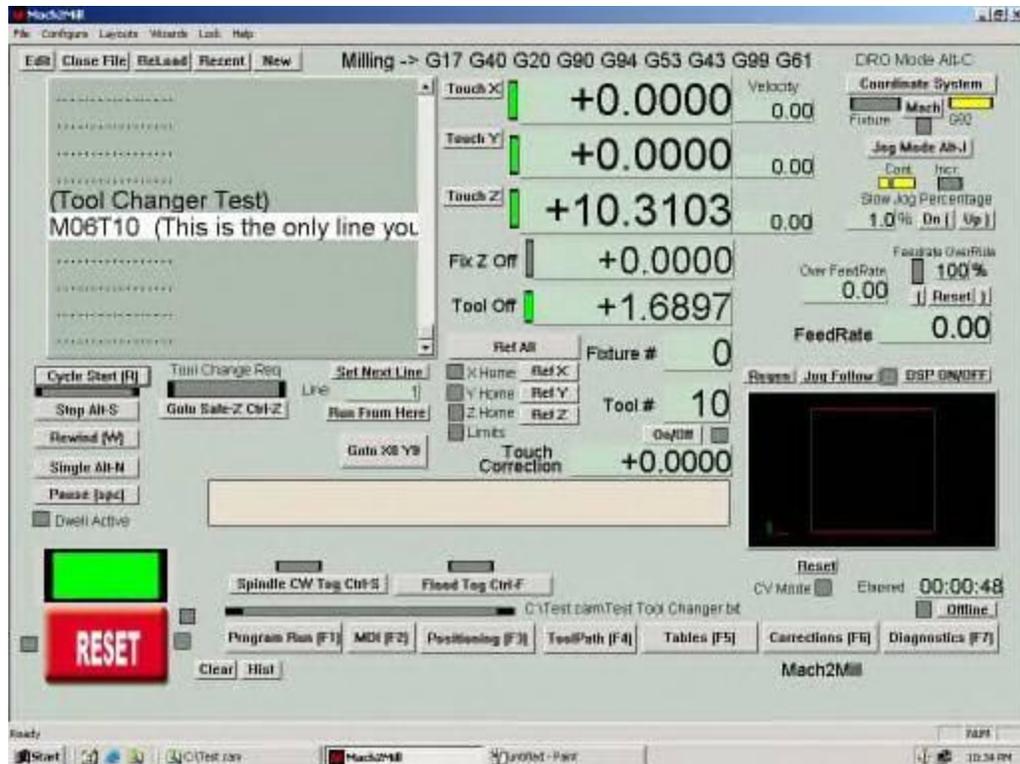
- Z is at 13.75
- Tool Off is actually off (not green) and set 0.00
- Tool # box is 0



Run the program

Here's what happens:

- The mill head moves to MACHINE Coordinates Safe Z (12.00)
- A small dialog box pops up with a number 0 in it.
- Although the Elapsed clock is not moving it is still keeping time and will update to when you hit OK.



Hit **OK** (don't change the 0 in it, just hit OK) and here's what you get

Things to notice:

- Z is at 10.3103, but the head did NOT move.
- Tool Off is actually on (green) and set 1.6897
- Tool # box is 10 (whatever number you used)
- If there are more lines in the program, the program **WILL** continue on.

Why is Z 10.3103?

From part 2 of this series you remember

When the Tool Height Offset is applied it **SUBTRACTS** the height of the tool from the Reference Line Height and puts the resulting value into the DRO (numbers on the screen).

$$12.0000 \text{ (Reference line)} - 1.6897 \text{ (Tool Height)} = 10.3103$$

Congratulations: You have now written your first program to use the Manual Automatic Tool Change.

Using It For Real

Here is a snippet of one of my programs

(Step 1)
(Spot Harsh Cover Holes)
M06T10
G00 Z1.0000

G81 X14.9116 Y1.4974 Z.70 R.80 F10.0
X14.9116 Y2.4697
X15.7071 Y3.3536
X16.3598 Y3.2652
X16.6352 Y4.0165
X17.2097 Y3.6187
X17.6075 Y3.0442
X16.8902 Y2.7348
X16.9445 Y2.1161
X16.2374 Y1.4090
G80

(Move Table to REAR for tool change)
G00 Z1.75
G00 Y0.2

(*****)

(Step 2)
(Drill 6-32 Pilot Holes)
M06T16
G00 Z1.0000

G83 X14.9116 Y1.4974 Z-.1 R0.85 Q.10 F3
X14.9116 Y2.4697
X15.7071 Y3.3536
X16.3598 Y3.2652
X16.6352 Y4.0165
X17.2097 Y3.6187
X17.6075 Y3.0442
X16.8902 Y2.7348
X16.9445 Y2.1161
X16.2374 Y1.4090
G80

(Move Table to REAR for tool change)
G00 Z1.75
G00 Y0.2

(*****)

(Step 3)
(Drill 5/16 Pilot Holes)
M06T13
G00 Z1.0000

G83 X1.1250 Y1.7500 Z-.1 R0.85 Q.35 F8
X1.1250 Y4.2500
X3.3750 Y4.0000
X2.5090 Y3.5000
X2.5090 Y2.5000
X3.3750 Y2.0000
X4.2410 Y2.5000
X4.2410 Y3.5000
X8.8750 Y4.2500
X8.8750 Y1.7500
G80

(Move Table to REAR for tool change)
G00 Z1.75
G00 Y0.2

(*****)

And so on...

What happens if the machine starts out and asks for tool #10 (spotter) and zips down to spot a bunch of holes. When it finishes the holes it moves the table to the back and out of the way. M06T16 comes a long and we begin the tool change sequence, we swap the bit, hit OK and the program starts drilling holes. Again we move the table out of the way, and start another change cycle, and drill more holes.

Final Thoughts

The sequence probably could use some cleaning up; leaving the number in the dialog box doesn't thrill me, not to mention a few other things. But setting it up is fairly quick and doesn't require a PHD. The more bells and whistles, the harder the setup.

I've been thinking about revisiting this and adding a few features to the entire series

- Auto tool height sensor
- Auto Spindle shutdown during the cycle
- Auto Coolant shutdown during the cycle
- Auto table positioning (move table to the rear)
- And a few other things.

I honestly don't know when I'll get to them; I think the Auto Tool Height sensor is coming first, maybe in a month or so.

