

Tool and zero setting and using Tool offsets within a program

Manual Tool Measurement

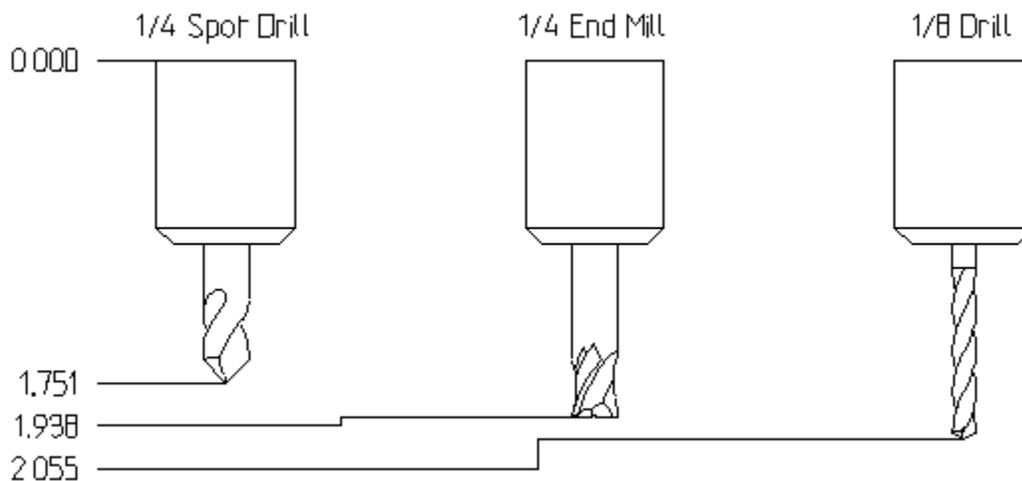
Tool offsets are one of the functions of DeskCNC that can aid in making multiple parts with less work. Let's say that you have 3 tools of different lengths. A spot drill, a drill, and an end mill.

The drill and the spot drill are usually held in a drill chuck, and the end mill can be held in a spring collet. These tool holders all mount the tool in a different Z position each time the tool is mounted. When using these tools, the CNC operator will measure each tool, each time it is used.

An alternative is to use solid tool holders with positive Z location. This can be accomplished with two drill chucks, and a solid end mill holder. On some machine spindles, quick-change collet holders are also available that enable the drills to be permanently located with respect to the Z0 of the machine. This eliminates the need to measure each tool every time they are used.

A typical part may have several holes spotted, then drilled, and then several contours milled with the end mill. If there are several parts to run, and if the part locating is complicated, it may be more economical to change tools between each milling operation and complete all operations on one part, before proceeding to the next part.

Fast and accurate tool change capability will make this even more useful.



If limit switches are installed on the machine, the Toolholder zero (called the Tool holder gage line), will be in the same location each time a given tool holder is installed in the machine. After the machine is homed, the machine coordinate system is used with the measured lengths shown above to determine the location of the tip of each tool installed in the spindle. With this method a Work offset is used to set the top of the material and the tool offset positions the tool tip to that relative location. The tool lengths can be measured in the machine, or frequently are measured with an offline gage at which the

Tool and zero setting and using Tool offsets within a program

lengths are measured and recorded. Then when the job is setup, the lengths of all the tools are entered in the tool offset table.



Click on the desired tool length and type in the measured length.

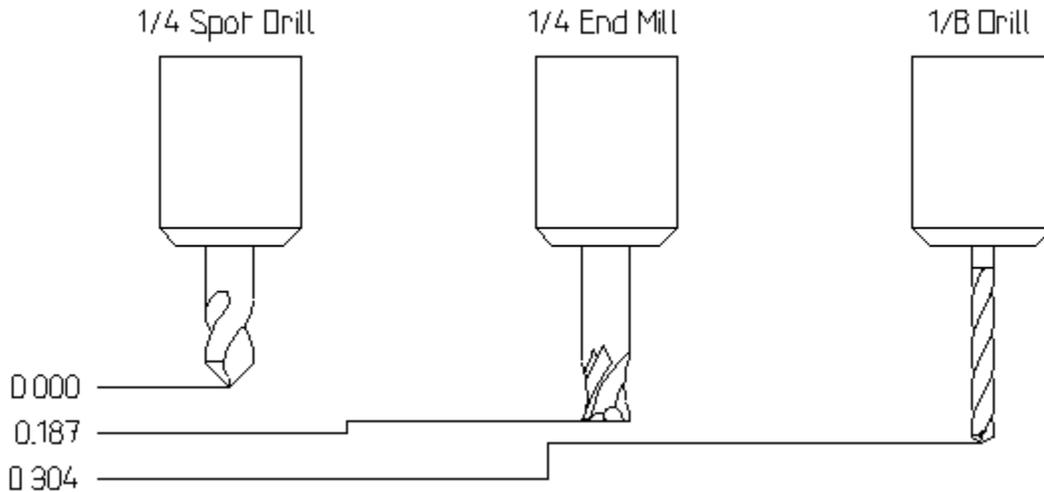
A screenshot of the 'Tool Library' dialog box. It contains a table with 7 columns: Slot, Id, Length, Diameter, Type, Ang/Rad, and Comment. The 'Length' column is highlighted. Below the table are buttons for 'Add Tool', 'Remove Tool', 'Print Table', and 'Save'.

Slot	Id	Length	Diameter	Type	Ang/Rad	Comment
1	1	1.5689	0.12500	End	0.00000	.125 End
2	2	1.00000	0.25000	End	0.00000	.25 End
3	3	1.00000	0.37500	End	0.00000	.375 End
4	4	1.00000	0.12500	Ball	0.00000	.125 Ball
5	5	1.00000	0.25000	Ball	0.00000	.25 Ball
6	6	1.00000	0.37500	Ball	0.00000	.375 Ball
7	7	1.00000	0.12500	Vee	20.00000	20 Degree
8	8	1.00000	6.00000	End	0.00000	6 MM End
9	9	1.00000	0.03125	End	0.00000	1/32" End
10	10	1.00000	0.18700	Ball	0.00000	.187 Ball

Click Save when done to save your tool measurement in the Tool table file: DeskCNC.tbl. If using homing switches, the tool lengths will usually be measured from machine zero at the top of the Z-axis travel. In this case the tool lengths will be negative numbers, measured “down” from the Z-0 position.

Since many hobby cnc machines do not make use of home switches, another technique is often used to keep track of the relative lengths of the tools in their holders. With this method, the first tool is measured and set to either a zero length, or else it is contacted against a gage length at a known location on the table. A 1-2-3 block can be used to touch off the first tool and set the coordinate zero point. Each subsequent tool is then measured, relative to the first and only the differences in length are recorded.

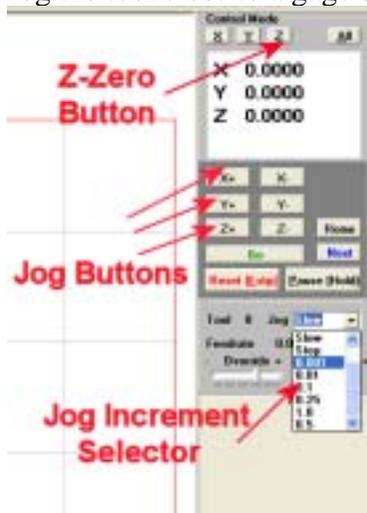
Tool and zero setting and using Tool offsets within a program



The problem with using this technique is that if the first tool has wear, or needs to be replaced, all the relative measurements are lost. For this reason, a tool holder with a dowel or other flat surface is often reserved for the zero setter (or standard) and kept in a safe place to maintain the relative settings of all subsequent tools. It is also possible to set the initial zero point from the spindle nose, provided it can be jogged into contact with the target gage surface.

To measure the first tool and set a relative zero:

Install the tool holder with the tool in the spindle.
Jog the tool close to a gage surface.



Switch to .010, then .001 increments as the tool gets closer to the contact surface. Each click of the jog button in the jog increment mode will move the jogged axis the specified distance.

When contact is made, click the Z-Zero button to zero the tool.
Enter 0.000 in the tool table for this tool.

Tool and zero setting and using Tool offsets within a program

Now jog the tool to the Z-zero of the work piece (usually the top of the stock)

Once again jogging with smaller increments until contact is made.

Record the distance from the Gage position to the Work piece Z0.

This is the Z value for the work offset, and can be recorded in the work-offset table (G54 for example).

In many cases, a satisfactory accuracy for tool setting can be obtained by placing a piece of paper between the tool tip and the gage surface, or top of the workpiece. As the tool approaches at .001 inch increments, at some point the paper can no longer be wiggled, and the tool has pinched it sufficiently to hold it in place. This is usually within .001-inch repeatability, and can be very reliable if an accurate paper stock is used. Often in commercial machine shops, a supply of cigarette papers is maintained. This paper is very thin and uniform (and inexpensive) and will yield an excellent tool setting capability. It is important when using fragile/brittle tools that contact with hard work surfaces is minimized. Contacting a polycrystalline diamond, ceramic, or carbide tool tip to a flexible paper surface is much less likely to fracture the edges or tip of the tool, than touching a hardened steel, 1-2-3 block.

Now install the second tool.

Jog to the gage surface and record the measurement. It will be the distance from the first tool. (.187 in the drawing above)

Enter this value in the tool table for Tool 2. (Setup-Tool Library)

Complete these steps for each tool.

When a multi-tool program is run, it will look like this:

N100 T01 M06 *Calls for tool 1 and changes to that tool number*

N110 G43 H01 *Loads tool offset 1, changes the machine position to reflect the installation of tool 1*

N115 G01 Z0 F10 *Moves the 1st tool tip to the top of the part (Z=0)*

N120 G01 Z.5

N200 T02 M06 *Calls for tool 2 and changes to that tool number*

N210 G43 H02 *Loads tool offset 2, changes the machine position to reflect the installation of tool 2*

N115 G01 Z0 F10 *Moves the 2nd tool tip to the top of the part (Z=0)*

N120 G01 Z.5

The programming process can be simplified by adding the G43 Hxx commands into the Tool change macros, contained within the TOOLCHANGE.CFG file.

Automatic tool measurement

With the DeskCNC TP-100 tool setter/probe, tool measurements can be automated. The tool setter is a spring-loaded switch that will record the tool lengths within the machine resolution, as the switch opens from tool contact.

Tool and zero setting and using Tool offsets within a program

There are two commands that can be used:

M96 – Moves in X and Y directions, to the locations specified under the tool sensor location



The tool sensor location is X=1.0 and Y=3.5 in this case



After the X and Y axis motion is complete, the Z-axis is moved down toward the sensor to record the tool length. The tool setter must be positioned in the specified X-Y position before the M96 command is issued.

M97 – Only the Z movement occurs with the M97 command. The tool setter must be positioned beneath the tool before the M97 command is issued.

In order for the tool measurement functions to operate correctly, the tool change macros must be correctly setup:

Tool and zero setting and using Tool offsets within a program

Following is a sample TOOLCHANGE.CFG file that will work for tool setting up to 6 tools, the file text starts after the dashed line:

```
-----  
[TOOL0]  
PICKUP0=M5  
PICKUP1=G49  
PLACE0=M5  
PLACE1=G49  
  
[TOOL1]  
PICKUP0=M5  
PICKUP1=G49  
PLACE0=M5  
PLACE1=G43 H01  
  
[TOOL2]  
PICKUP0=M5  
PICKUP1=G49  
PLACE0=M5  
PLACE1=G43 H02  
  
[TOOL3]  
PICKUP0=M5  
PICKUP1=G49  
PLACE0=M5  
PLACE1=G43 H03  
  
[TOOL4]  
PICKUP0=M5  
PICKUP1=G49  
PLACE0=M5  
PLACE1=G43 H04  
  
[TOOL5]  
PICKUP0=M5  
PICKUP1=G49  
PLACE0=M5  
PLACE1=G43 H05  
  
[TOOL6]  
PICKUP0=M5  
PICKUP1=G49  
PLACE0=M5  
PLACE1=G43 H06  
  
[Sensor]  
SensorHeight=1  
LocationX=0  
LocationY=0  
-----
```

File text ends before the dashed line.

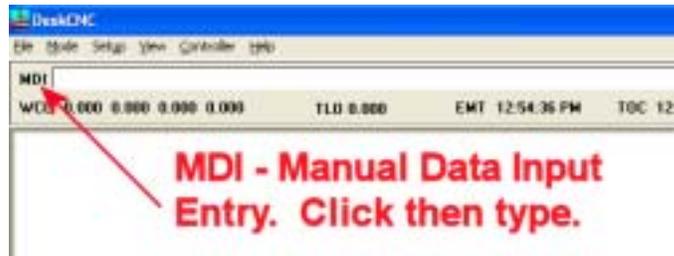
M5 is a spindle stop command and will stop spindle rotation if spindle relays are connected. If spindle relays are not used, be sure to stop the spindle manually before attempting to measure a tool length. Note that the TOOL0 contains a G49 command, G49 cancel any existing tool offsets.

Tool and zero setting and using Tool offsets within a program

To measure a tool with the tool setter, first measure the height of the tool setter.

- 1) Set the sensor height to .0001 in the Setup-Machine setup-Tool Sensor/Probe
- 2) Set tool 1 length to 0.000 in the tool table (Setup-Tool Library)

In the MDI command line:



- 3) Type "T1 M6" in the MDI box, then press Enter.
The tool indicator, next to the Jog increment drop down will show tool #1.
- 4) Load the first tool in the spindle.
- 5) Jog the tool down to the base plane of the Tool setter.

This can be the mill tabletop, or the top of the part. Use a piece of paper to locate the tool within a .001 or .002 inch of this surface.

- 6) Zero the Z-axis by pressing the Z-zero button.
- 7) Jog the tool up in Z to clear the tool setter
- 8) Place the tool setter beneath the tool. Make sure it is plugged into the controller.
- 9) Type "M97 F5.0" in the MDI box then press Enter.

The tool will move downward in Z and contact the tool setter. The tool offset in the tool table will update and the tool offset will become active. (The same effect, as if G43 H01 had been entered). The tool will move in the plus Z direction a few steps, then may pause, or may rapid up .75 inch and stop. If it does not rapid in Z, use the Z+ jog button to move away from the Tool setter.

At this time the Tool-1 offset in the tool library contains the height of the tool setter.

- 10) Transcribe this number to the Setup-Machine Setup-Tool Sensor/Probe, Sensor Height value.
- 11) Save the new machine setup settings.

After the controller restarts you can measure all your tool lengths.

To Measure tool lengths after Tool-1 has been used to set the Z-zero reference plane:

- 1) Set all tool length offsets to 0.000 in the tool library

Tool and zero setting and using Tool offsets within a program

- 2) Jog Tool-1 down to the setter reference plane and use a piece of paper to contact the surface with the tool
- 3) Set Z=0 with the Z-zero button.
- 4) Jog the tool plus in Z to clear the tool setter, and place the setter on the zero plane.
- 5) Enter "T2M6" in the MDI box and press enter.
- 6) Load Tool 2 in the spindle
- 7) Enter M97F5 in the MDI box and press enter

Tool 2 will feed down to the tool setter and the relative length will be measured and recorded in the tool2 length offset of the tool library.

Repeat steps 5-7 for each tool.

To test the tool offsets, the following program will load each tool and feed it to Z=0 with its tool offset in effect(make sure that t01 has been used to set the Z=0 reference plane first):

```
N100 T1M6  
N105 G01 Z0.0 F10.0  
N110 M02
```

After all the tools are measured, click the SAVE button on the Tool table dialog (see page 2), to save measured tool lengths for use in future sessions of DeskCNC.