Basic Digital Read-Out Functionality on a Mill

By R. G. Sparber

Copyleft protects this document[1].

There is a running joke among owners of Digital Read-Out (DRO) – What is the cleanest thing in any shop? The "FUNCTION" button on your DRO. This article focuses on the everyday functions found in most DROs rather than those advanced functions that are rarely if ever used.
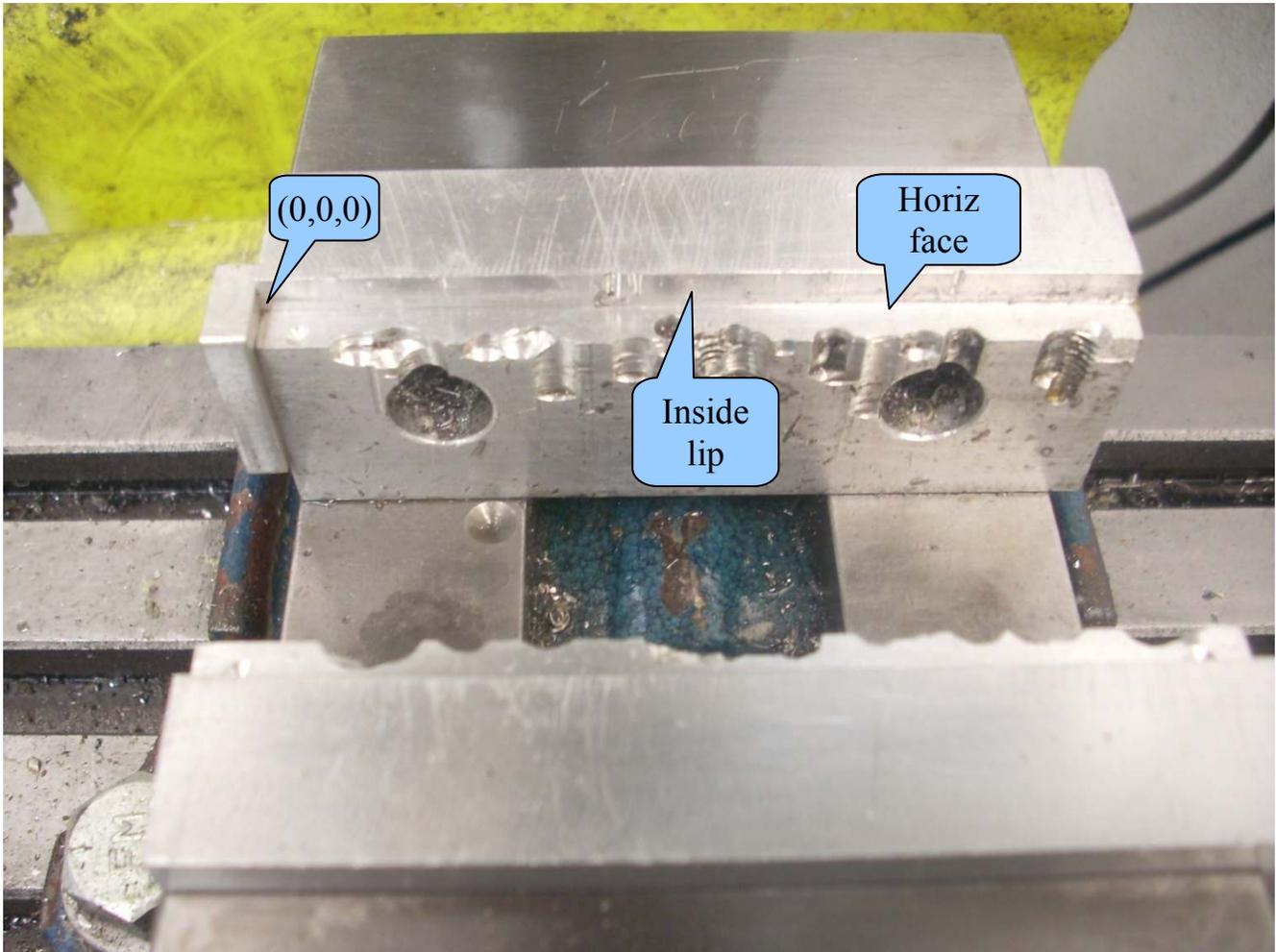
Contents

---

[1] You are free to copy and distribute this document but not change it.

Absolute Zero

We all have to start someplace. The mill has X, Y, and Z axes but we must define what we mean by zero on each one.


I know these "soft" jaws[2] look rather ratty but they are, in fact, very precisely aligned with the X and Z axis. The back jaw is fixed and the front jaw can move. Attached to the left side of the back jaw is a stop. I usually want to define X = 0 as the right face of this stop.

There are two ways to define Y = 0. The CNC standard is to set Y = 0 at the inside lip of the movable jaw. Whenever a new sized part is clamped into the vise, one must touch down on the reference surface to define Y = 0.

I prefer to define Y = 0 against the inside lip of the fixed jaw. In this way I don't have to keep touching down on my reference surface in order to find Y = 0. Similarly, I set Z = 0 on the horizontal face of the lip.

---

2   http://rick.sparber.org/Articles/sj/sj6.pdf
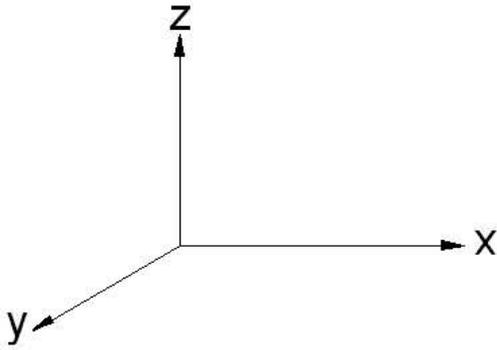
The intersection of my X, Y, and Z axes is my (0,0,0) reference point which is also called the origin.

Note in this graph that X becomes larger as I move away from the origin to the right. Y becomes larger as I move to the front. Z becomes larger as I move up.

After cleaning off all mating surfaces, I have clamped a precision 1-2-3 block into my vise. This is for instructional purposes only. I have no intension of touching this hardened block with a cutter.

The block is snug against the fixed jaw's lip and the fixed jaw's X axis stop. Therefore, the back left, bottom corner of the block is at my origin.

I have put a "spud" into my drill chuck as a means of locating surfaces. This is not particularly accurate but is easier to understand than if I used my Electronic Edge Finder. I'll get to the EEF later.

In order to set Absolute Zero on the X axis, I start by moving the Y axis such that I'm away from the corner. The tip of the spud is then lined up with the left face of the 1-2-3 block. I then hit the ZERO button for the X axis.
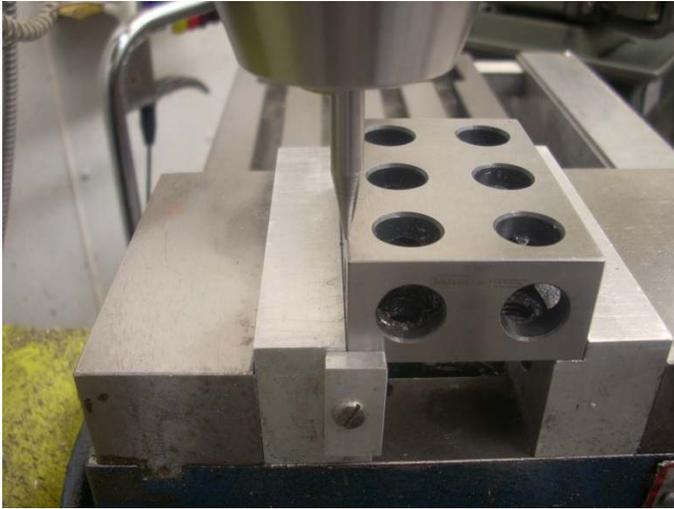
Next I move to the back of the 1-2-3 block again staying away from the corner. When the tip of the spud is lined up with the back face of the block, I hit the ZERO button associated with the Y axis.

Why stay away from the corner? Because it is hard for me to accurately sight both axes at the time. If using an EEF, it is essential you are completely on a flat surface so away from the corner.

I can zero my Z axis display too but have not defined the location yet. That is a function of the tool I mount in my spindle. Once the tool is mounted, I can touch the end of it down on the horizontal face of my fixed jaw and set Z = 0.
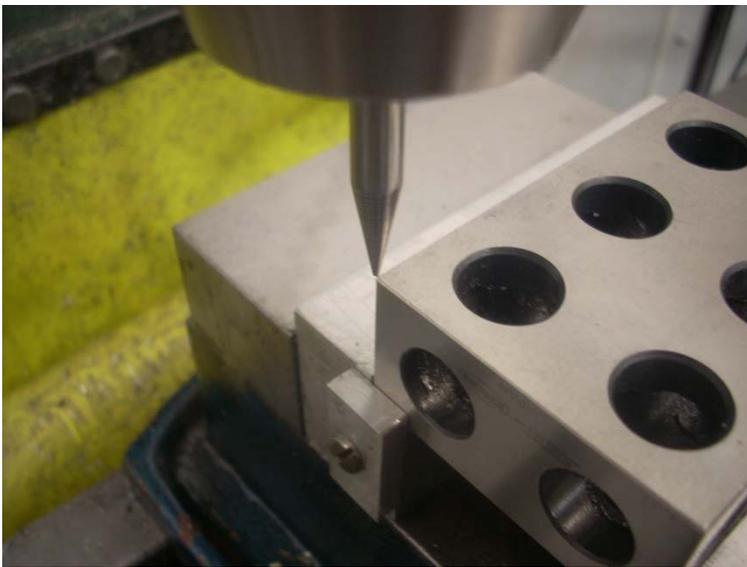
To recap, I have defined X = 0 at the left face and Y = 0 at the back face of my 1-2-3 block.

One very confusing thing about setting up my DRO was "Polarity". With the back left corner of the block defined as my origin, I want the numbers to increase as I move my **cutter** to the right and/or to the front. But wait, the cutter isn't really moving! The part is moving.

My solution was to ignore reality and think of it as the cutter moving while the part stands still. This bit of fiction has served me well over the years.

Oh yes, one more thing. I have labeled each of my axis hand cranks with a "POS" and "NEG" rotational direction in hopes of minimizing the number of times I move the cutter in the wrong direction and spoil my project. It has saved me a few times.
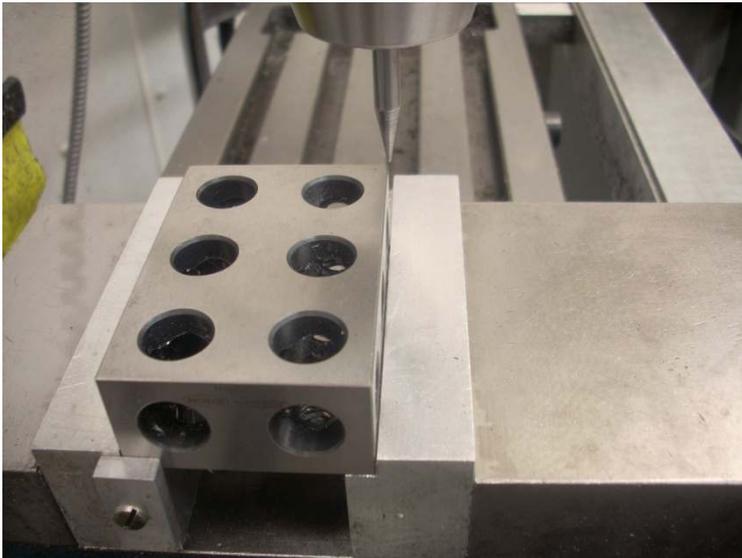
I have now cranked the X axis so my spud is 3.000" to the right of my origin and my Y axis 2.000". These are the exact value on the display. Please do not confuse the precision of the numbers with the absolute accuracy of the DRO. That is a different can of worms. See your DRO user's guide for details.

The lower left corner of my 1-2-3 block is my origin. My spud is at X = 3.000", Y = 2.000".

In many cases the exact location of the origin is not that important. What often matters is that the relative position of machined features be precise. As long as the origin does not move, everything works out fine. If you must have the origin precisely aligned with two reference faces, a Dial Test Indicator or EEF can be employed.
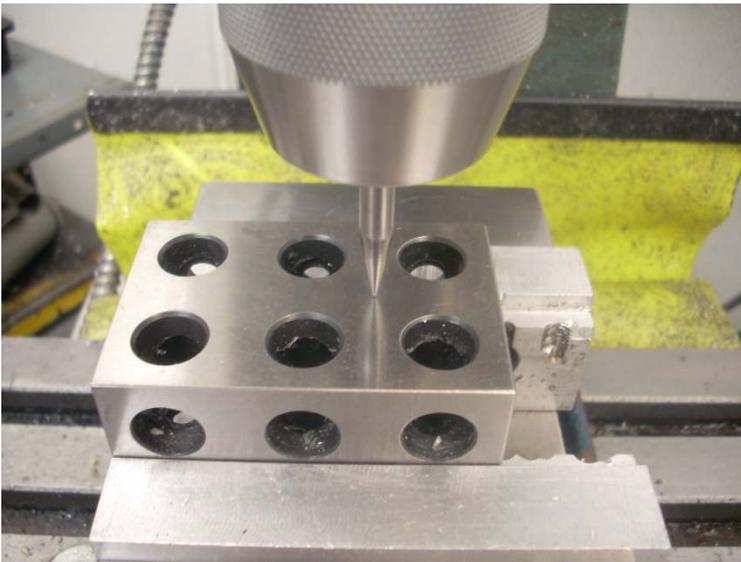
Incremental Zero

There are times when some machined features must be precisely set relative to our origin while other times I must be a precise distance from a different point. The latter case is when defining an incremental zero is handy.

For example, say I want to define the absolute location (3.000", 2.000") seen here as an incremental origin. I first push the ABS/INCR button such that the INCR indicator lights. Then, without moving either axis, I hit the Zero button on both the X and Y scales.

I now have my absolute (0,0) point at the lower left corner of my 1-2-3 block and my incremental (0,0) point at the upper right corner. Just by pushing the ABS/INCR button I am able to switch between these reference points.

While in INCR mode, I have moved 1.000" to the left of my INCR X = 0 point and also 1.000" in from my INCR Y = 0 point.

Remember how I set up my Polarity? Moving my spud to the right or towards me is positive. You can see here that my spud is to the left and back from the front right corner. This is why both X and Y are negative with respect to my INCR origin which is the front right corner.

If I switched back to ABS, the display would read X = 2.000" and Y = 1.000".

Tool Offset Define and Tool Offset Use

Using the X and Y axes, we can accurately locate at any point. Then what? If we want to drill a hole, we just chuck up the drill and make some chips. But what if we want to mill a feature?

to center

to cut

Top View

Side View
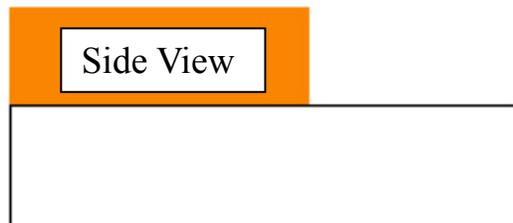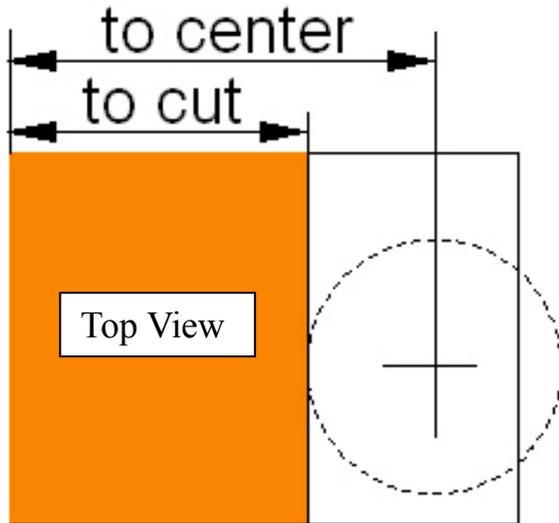
If we just start cutting with an end mill, there might be a surprise in store. The center of our end mill will be at the chosen location but the step we cut will be at a distance equal to half of the end mill's diameter. What do we do if we want the step to be at the defined location? The answer is to use the Tool Offset functions.

Assume that my origin is at the upper left corner of the block. With no tool offset set, the DRO will show me the distance from the origin to the center of the end mill. What I want is to know the distance from the origin to the "face" of the end mill that will cut my step. In this example, I am cutting with the left face of the cutter. This means that I want the DRO to subtract half of the diameter of the cutter from its previous reading. Then it will display the distance from the origin to the cutting face.

If I wanted to cut on the right face of the end mill, I would tell it so and the display would then equal the original value plus half of the diameter of the cutter. The same goes for cutting on the front or back face of the end mill modifying the Y axis.

If you buy new end mills, then it is very likely that the diameter marked on the cutter will be the true diameter. If you got a good deal on resharpened end mills, it would be wise to measure their diameter. I measure them by cutting a 0.1" deep slot in a piece of scrap aluminum. The width of that slot tells me the cutter's effective diameter. Any run out in the cutter or holder will be included in this measurement.

I maintain a table showing tool number and diameter which is stuck to the front of my mill. One of the tool diameters is for my Electronic Edge Finder and the rest are cutters.

Let's run through an example. Say I have the block of metal shown above and I want to cut a step at a distance of 2.000" from my origin. To do this I will use my brand new 5/8" end mill. Since it is new, I will assume it has a diameter of 0.625". It will be my tool number 2 which I define with my Tool Offset Define function.

The next step is to use the newly defined cutter. I select the Tool Offset function. The cutter's offset is now loaded and waiting to be used. I have 5 choices:

<div align="center">rear</div>

<div align="center">left   no offset   right</div>

<div align="center">front</div>

Since I will be cutting on the left face, I select "left". If later I want to remove the offset, I select "no offset".

You can imagine milling a pocket and need to change offsets as you go around the hole. First you cut the left wall so select "left". Then you cut the back wall so before moving into position, so select "rear". Then you select "right" to cut the right wall. And finally, you select "front" to cut the front wall. Finish by selecting "no offset" so you remove the offset. Sure is handy!

Electronic Edge Finder

Some DROs have an Electronic Edge Finder function. The EEF is mounted into the spindle and is used in conjunction with the DRO to automatically set zero for the X or Y axis.

You crank the table until the EEF touches the part. This causes an electrical circuit to complete and tells the DRO to zero the selected axis. I always back up and verify it got the right point. Any tiny bit of swarf will cause error here. So will any insulating material like oxide or oil. Usually two tries is enough to give a very accurate zero set.

Don't forget our old friend "Tool Offset". We will be touching down on one of the faces of the EEF so will need to tell the DRO to compensate. My EEF tip is 0.200" in

diameter and I have it defined as tool 9 so it is easy for me to remember.
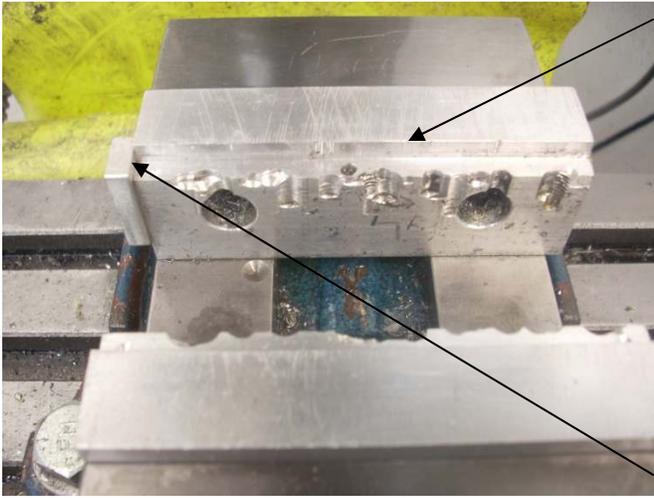
Say I plan to pick up the rear jaw's lip as my Y = 0 point. I would go to Tool Offset and pick tool number 9. Then I would set "back" to tell the DRO I was going to use the rear face of the tool. I would then hit ZERO on my Y axis to define which axis will be zeroed at touch down. I then crank the Y axis feed until the EEF touched my reference surface. This zeros the Y axis display.

Normally I will set up my X = 0 point at the same time so just need to set "left" and touch down with the left face of the EEF.

Acknowledgements

Thanks to Jim of the yahoo group mill_drill for suggesting this topic.

Rick Sparber
rgsparber@aol.com
rick.sparber.org