The Unofficial Shumatech DRO-550 Application Notes for the Mill, Version 2.0

By R. G. Sparber

November 25, 2009

Disclaimer: Use these notes at your own risk. Although the author has made every effort to produce an accurate document, he assumes no responsibility for errors.

Revision History

Version	description
1.0.0	The content of the “Unofficial Guide to the DRO-350” was used as a starting point for creating this guide.
1.0.1	Typos and spelling errors corrected with help from Larry Gill. Grid Hole Pattern explanation added.
1.0.2	Added examples for Grid hole pattern
2.0	Changed to “unofficial”

1 You may freely distribute this document but not change it.

Table of Contents

Scope

This document is meant to work in conjunction with the “Shumatech DRO-550 User's Guide for Software Release 1, Version 1”.

Background

In “Shumatech DRO-550 User's Guide for Software Release 1, Version 1” you will find which buttons to push in order to enable the multitude of available functions. What is not present is an explanation of ***why*** you would want to push those buttons. I hope to supply at least some of the ways these functions are useful in my shop. I am presenting information related to using the DRO on a mill. At some future date I hope to create a similar document devoted to using the DRO on a lathe.

Labeling Convention

The DRO-550 has 3 large displays. You may see, for example,

Function
position
tool offset

Within the text part of this document, you would see this represented by

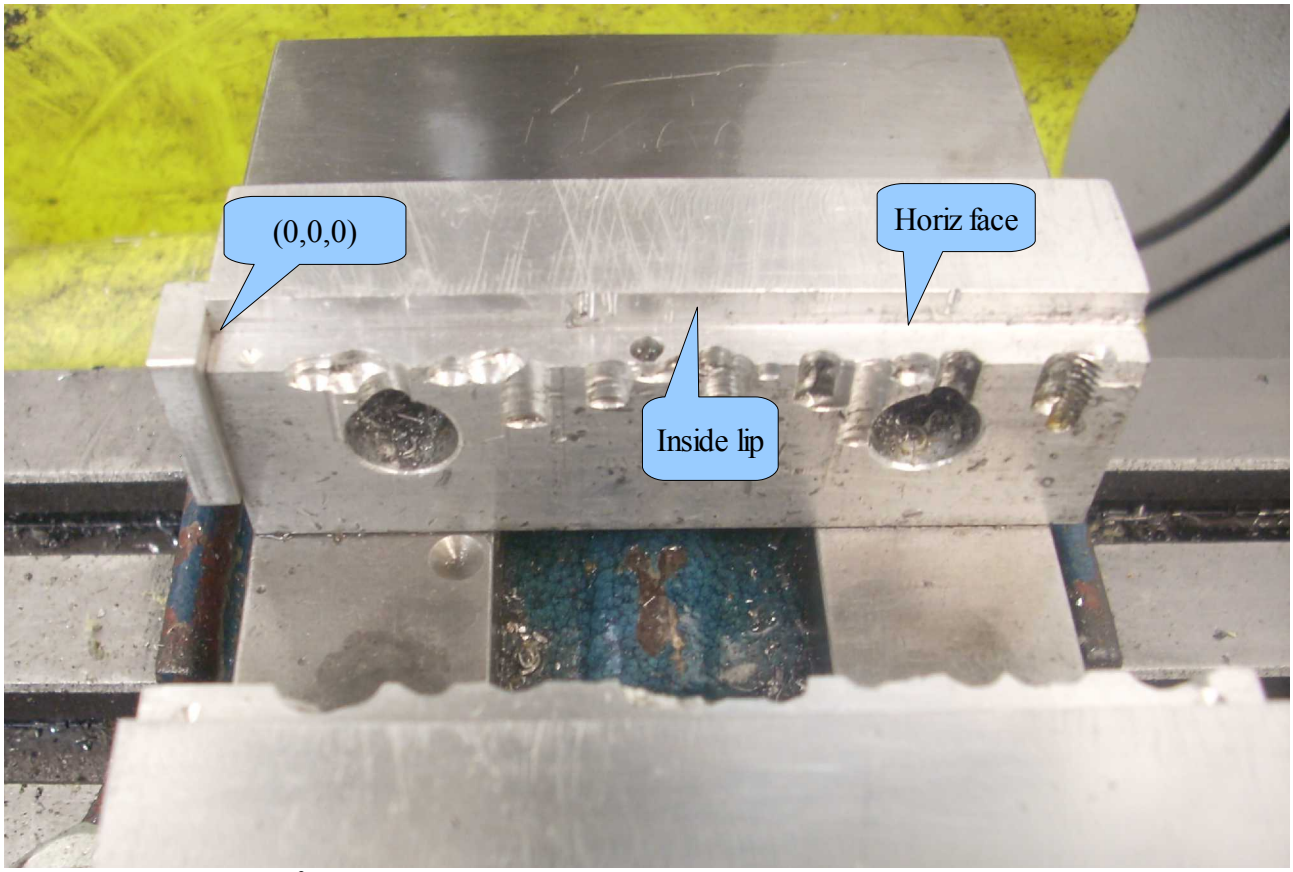
[Position [tool offset

Mill Functions

Absolute Zero

Absolute Zero via Zero or Enter Button

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

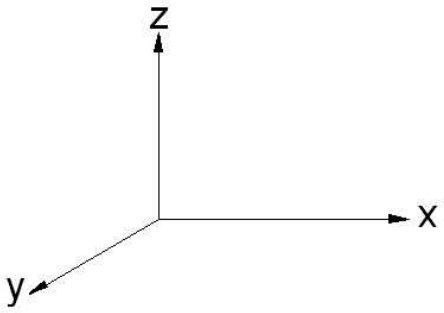


I know these "soft" jaws[2] look rather ratty but they are, in fact, very precisely aligned with the X and Z axis. The back jaw is fixed and the front jaw can move. Attached to the left side of the back jaw is a stop. I usually want to define X = 0 as the right face of this stop.

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.

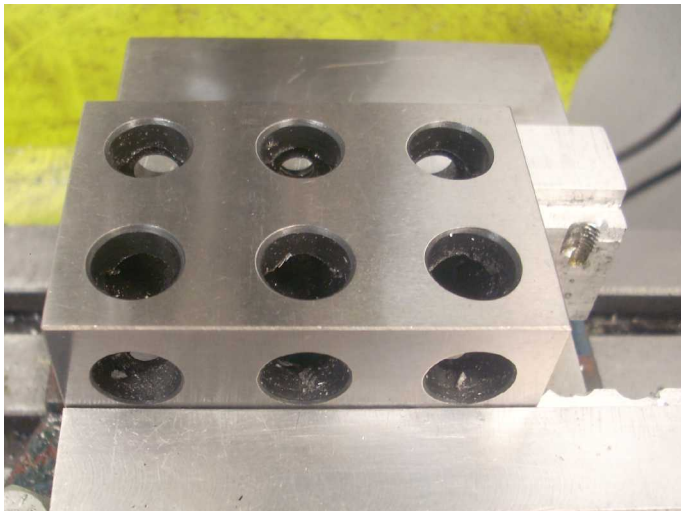
2 http://rick.sparber.org/Articles/sj/sj6.pdf

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.



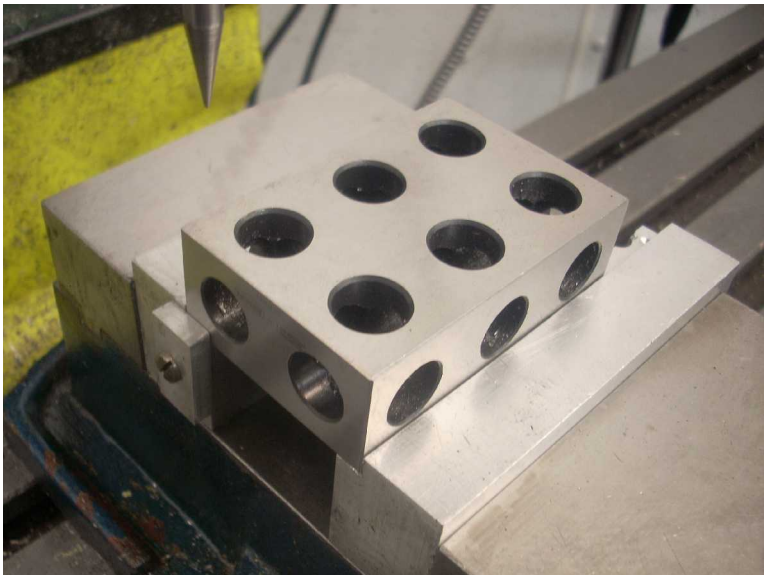
The intersection of my X, Y, and Z axes is my (0,0,0) reference point which is also called the origin.

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.

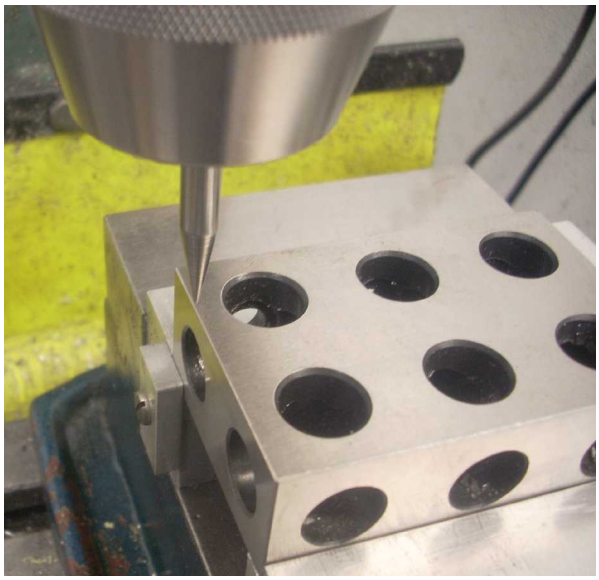


After cleaning off all mating surfaces, I have clamped a precision 1-2-3 block into my vise. This is for instructional purposes only. I have no intention of touching this hardened block with a cutter.

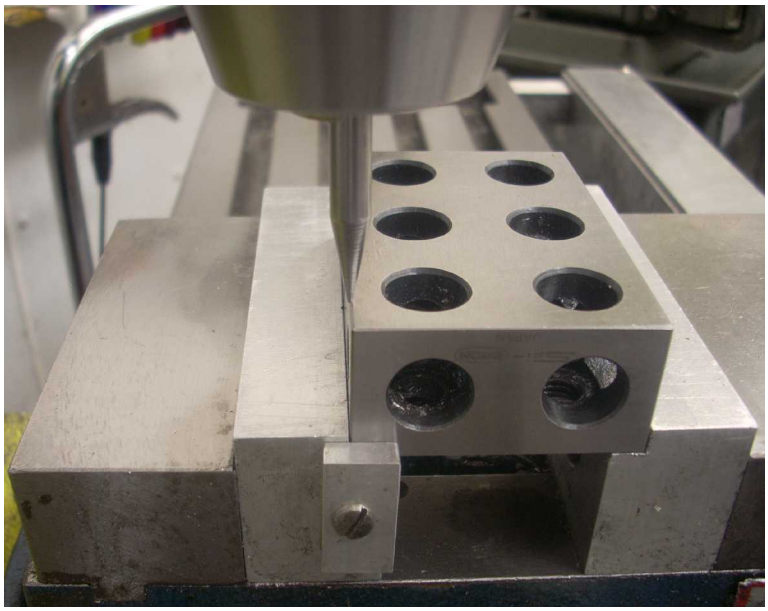
The block is snug against the fixed jaw's lip and the fixed jaw's X axis stop.



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 the with the left face of the 1-2-3 block. I then press the ZERO button for the X axis **twice**. The first push of an axis' zero button says I want to set a zero. The second push of that same zero button says I want to set zero at the present physical location. I could have pushed the ZERO button once and then used my EEF to actually set zero.



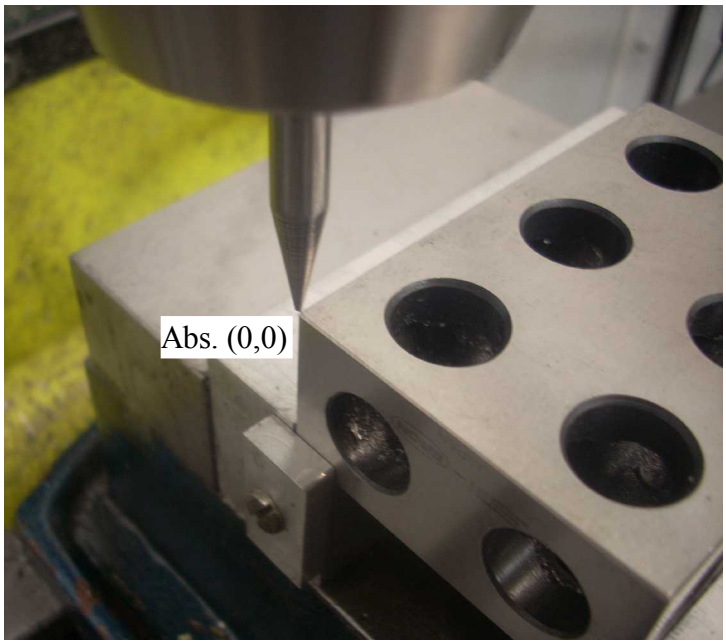
Next I move to the back of the 1-2-3 block again staying away from the corner. When the tip of the spud is lined up with the back face of the block, I press the ZERO button associated with the Y axis **twice**.

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.

Sorry about the glare, but you can hopefully see that I now have my X and Y axes zeroed. My X axis shows a negative sign in front of my zeros while the other two axes are positive zero. This is not important. Both mean zero.

If you are paying close attention, you will note that I have a lathe faceplate on my DRO but have pasted over the Y and Z buttons next to their displays. It is just not worth the trouble to pull off this faceplate and stick on the right one. Who knows, I may yet move this unit to my lathe.

I have zeroed 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. In all cases I have set zero by pushing the associated zero button **twice**.

The ability to define my (0,0) point in the back left corner is accomplished through the [Setup [axis and [Setup [scale functions. I found these concepts very confusing for a while. 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.

If you want to have your origin in the back, left corner, then set up your axes as

X – [reverse [no

Y – [reverse [yes

Z – [reverse [no

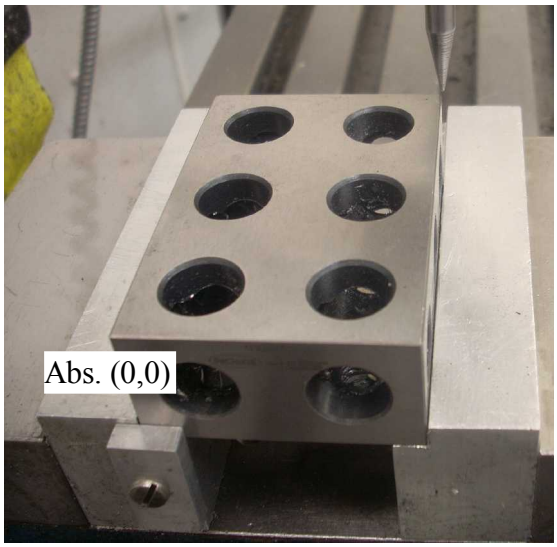
That "[reverse [yes" flips the polarity of the Y axis so that as the ***cutter*** moves to the front, the display shows an increasing number.

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 3.0005” to the right of my origin and my Y axis 2.0000” . These are the exact value on the display. Please do not confuse the precision of the numbers with with the absolute accuracy of the DRO. That is a different can of worms[3].

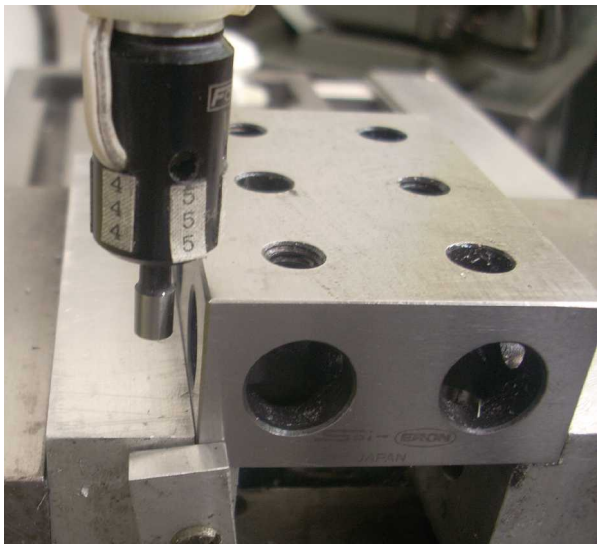
The lower left corner of my 1-2-3 block is my absolute origin. My spud is at X = 3.0005”, Y = 2.0000”.

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.



3 See many articles related to accuracy at http://rick.sparber.org/sh.htm.

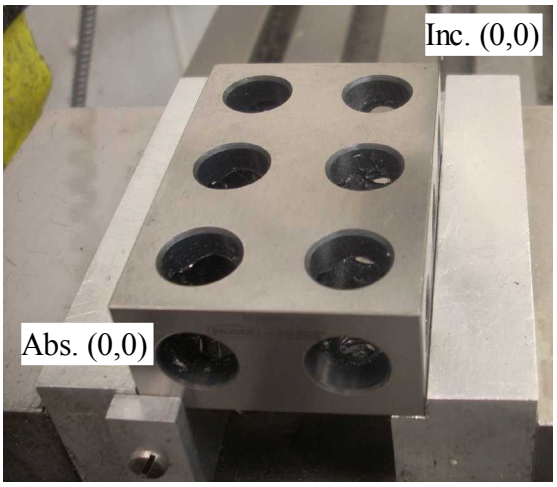
Absolute Zero set by an Electronic Edge Finder



An Electronic Edge Finder is just a probe that is insulated from the rest of the machine but connected to the DRO-550 via the Flex 1 or Flex 2 port. The probe is fitted in place of a cutter. When this probe comes in contact with a surface electrically connected to the machine, a circuit is made. This circuit can be used to tell the DRO that we just touched down. When used in combination with [Tool Offset , we can easily set Absolute Zero for a given axis on a given surface. More on my EEF can be found in the Appendix on page 32.

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.

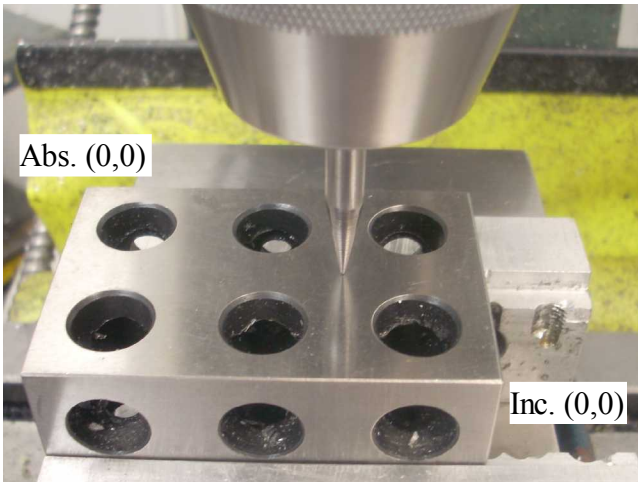


For example, say I want to define the absolute location (3.0005”, 2.0000”) seen above as an incremental origin. I first push the ABS/INCR button such that the INCR LED lights. Then, without moving either axis, I press the Zero button next to both the X and Y displays.

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 1.0005” to the left of my INCR X = 0 point and also 1.000” in from my INCR Y = 0 point.

Note that my INCR LED is lit.

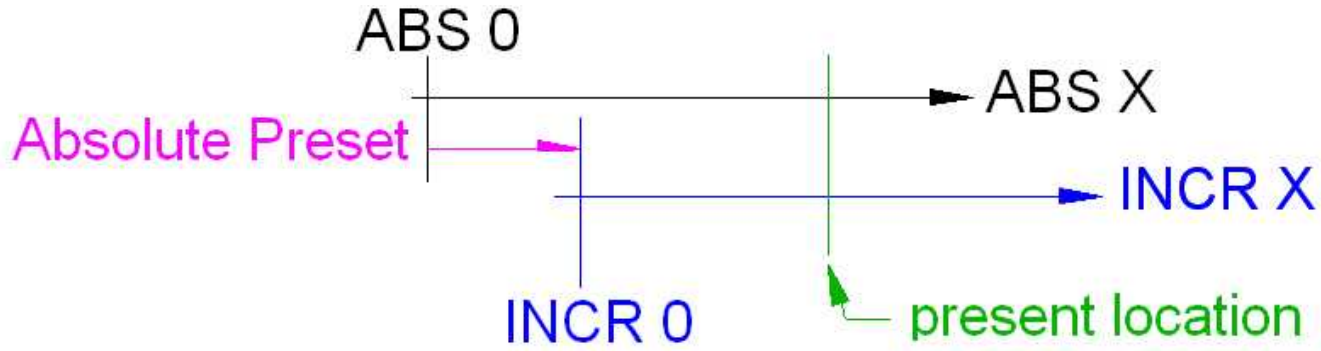


Remember how I set up my axes? Moving my spud to the right or towards me is positive. You can see here that my spud is to the left and back from the front right corner. This is why both X and Y are negative with respect to my INCR origin which is the front right corner.

If I switched back to ABS, the display would read X = 1.9995” and Y = 1.0000”. If INCR had been X = -1.0000” instead of -1.0005”, then ABS X would be 2.0000” rather than 1.9995”.

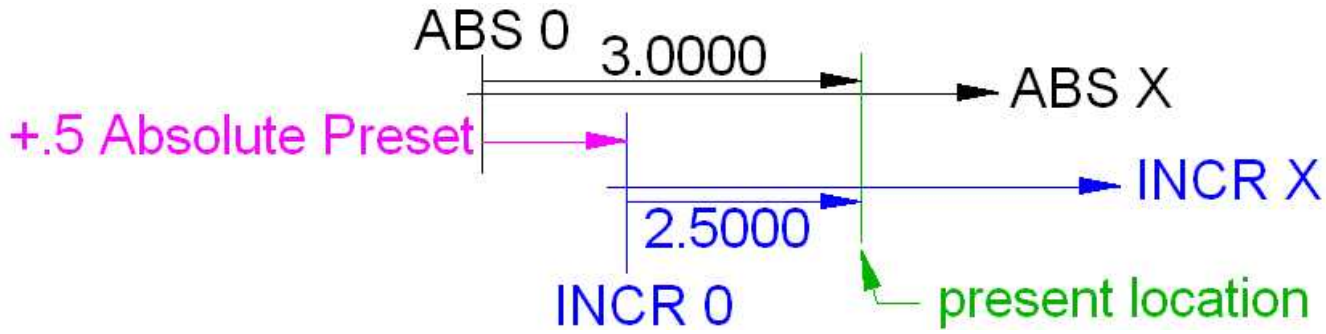
Absolute Preset

This one sure had me turned around for a long time. Start by thinking only about the absolute X axis and the incremental X axis.



On the top I have my ABS X axis with its ABS 0 set. Below it is the INCR X axis and its INCR 0 set. These two axes are normally unrelated. By using the ABS and INCR Zero functions, we can set both at any location we want. But what if we wanted to set a relationship between them? Say I want my INCR

X axis to be shifted by a fixed amount relative to my ABS X axis. This shift is accomplished with my Absolute Preset function. Let's move on to a concrete example.



Say I want my INCR X to be shifted 0.50000" up the ABS X scale. I would have input a +.5 Absolute Preset. I happen to be at an ABS X location of 3.0000". When Enter is press, my INCR X zero point will be set to a point lined up with an ABS X value of +0.50000". This means that when my ABS X value is 3.0000", my INCR X value will be 2.5000".

Now, lets do it again but this time looking at the DRO. I start out at an ABS X = 3.0000". I press the X preset button and input ".5" which displays at "0.5". Internally it will be stored as 0.50000".

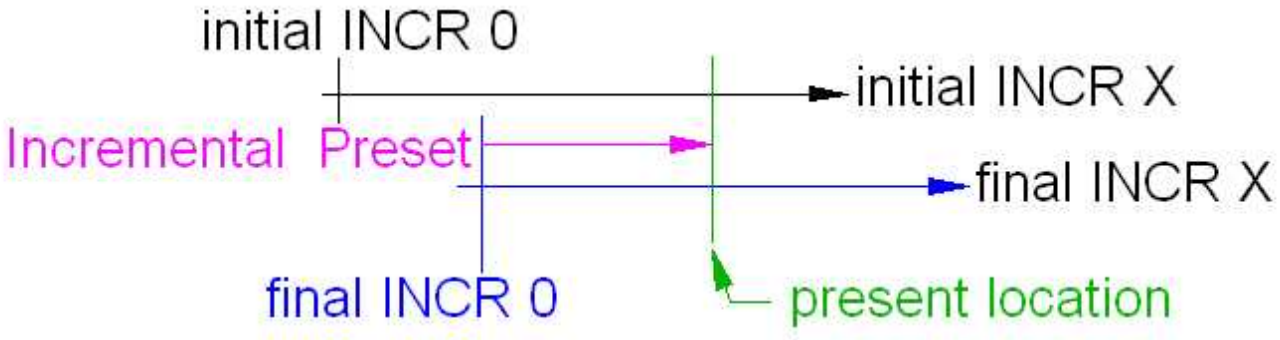
I then press the Enter button and we see:

You can see from the LEDs that we are now in INCR mode. Our INCR X value is now 2.5000". The INCR Y value just happened to be zero and INCR Z was 0.008". The change to the INCR X axis has no effect on the Y and Z axes.

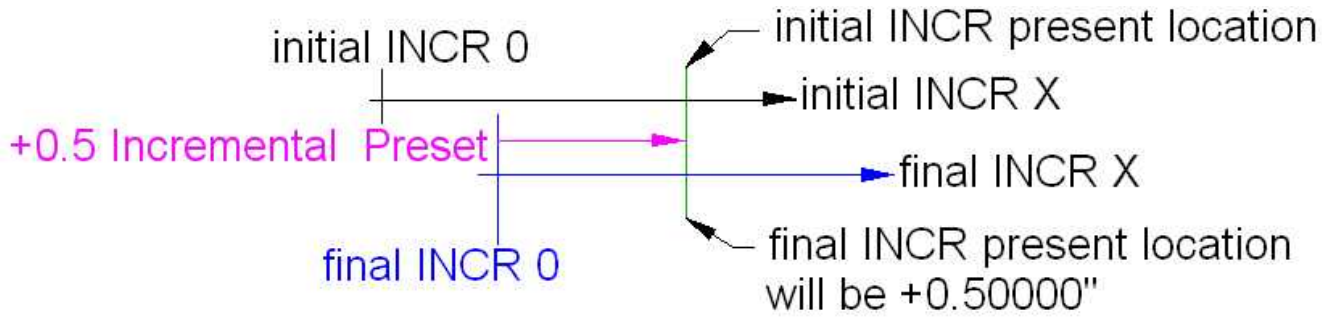
Hopefully I have explained to you what Absolute Preset does. Now, as to **why** you would use this function, I can't think of one right now. Hopefully others will read this article and help me out.

Incremental Preset

Similar to the Absolute Preset, Incremental Preset also changes only the Incremental zero point of an axis.



We have an initial INCR 0 set point which will be modified by the Incremental Preset to produce a final INCR 0 set point. Only the INCR axis is changed and the initial INCR 0 is lost.



Say I am sitting at some initial INCR position on the X axis. I want to define my INCR 0 to be some distance from my present location. If I wanted that distance to be 0, I would just use my Incremental Zero <10> function. But if I wanted my INCR 0 to be such that I was sitting at +0.5000", I would use my Incremental Preset function. I don't physically move my position. My INCR 0 moves. Kind of hard to get your mind around.

Let's look at a few screen shots.

I am in INCR mode as can be seen by the LED. My INCR Y display says I happen to be at 0.

Say I want to move my Y axis INCR 0 such that I am at +0.5000". I first press the Y preset button and then input .5. the display shows 0.5" but internally it will be stored as 0.50000".

When I press the Enter button, my Y display shows 0.50000". My mill table did not move. I just defined my INCR 0 for the Y axis to be 0.5000" in back of my present location.

Incremental Preset can define INCR 0 at any point within the range of the DRO relative to your present location.

Hopefully I have explained to you what Incremental Preset does. Now, as to **why** you would use this function, I can't think of one right now. Hopefully others will read this article and help me out.

One point of confusion is that you input an offset and after pressing Enter, the display shows the negative of this number. Some expect to see just the number. Here is one way to look at the logic.

I am at zero on a given axis.

```
----------------------------0------------------------------>+
```

I want to define a NEW zero point on this axis. Now, I am not physically moving the table. When I say I want my zero to move by +5 inches, picture the zero point being picked up and moved to +5 inches.

```
                            original
----------------------------0------------------------------>+
                        present position

                                  |---------5"--------->|
                                                      new
------------------------------|----------------------0-------->+
------------------------------|----------------------0-------->+
                               -5"                        | my new zero point
                         present position
```

my present position relative to my new zero point is -5. The display showing my present position as "-5.000" on it. When I have move the table so the display reads 0, I will have moved +5 inches.

Now, consider what it means if entering +5" causes the display to show +5".

```
                            original
----------------------------0------------------------------>+
                        present position

          <--------5"----------|

     new zero
------0------------------------------------------------------>+
```

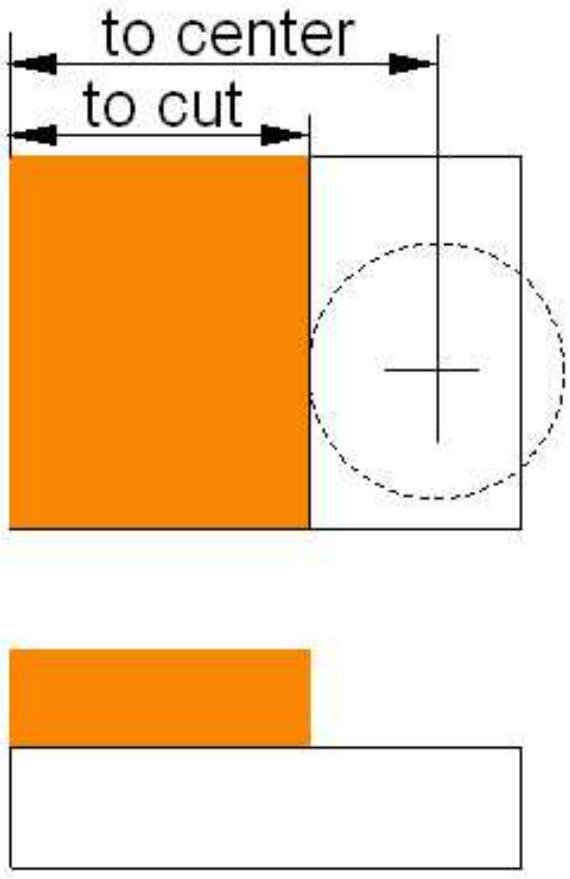
I must move my new zero to the left if I want my present position to show +5".

```
------|-----------------------|----------------------0-------->+
      | my new zero       +5"
                        present position
```

So in this case you entered +5 to move the zero point positive 5 inches and it actually moves it negative 5 inches. Is that what you really wanted to do?

[Define [Tool Offset and [Position [Tool Offset

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



If I want to mill the back side of this block, the front side of the end mill must move back by its radius. This movement puts the front side of the end mill in the right place but I must still tell the DRO that this is my new absolute (0,0) point. We do this temporary coordinate modification with the tool offset function. Unlike using the absolute zero set, I can change my tool offset with the push of a button.

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.

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 “left side” of the end mill which will cut my step. In this example, I am cutting with the left side 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 side.

If I wanted to cut on the right side 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 side of the end mill.

If you buy new end mills, 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. There is room for 16 tool diameters and, so far, this has been plenty. One of the tool diameters is for my Electronic Edge Finder and the rest are cutters.

Let's run through another example.

Say I have a block of metal and I want to cut a step on its back face. The back left corner of the block is at my absolute origin. 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. On a critical job, I will later make a trial cut and verify that this assumption is correct.

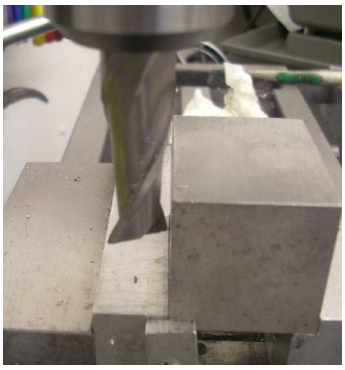
Using the shortcut, I push the FUNC and then the 5 buttons to call up the [Define [Tool Offset function. I then tell it to define tool number 2 by pressing the 2 button. The next prompt is for offset 1 which is the tool diameter. After entering .625" I press Enter. It will then prompt me for offset 2, the Z offset. I leave it at 0. Using the Z offset involves a class of machining that I normally don't worry about. Instead, I just touch down my cutter and set Z = 0. Press Enter and I am done.

The next step is to use the newly defined cutter. Again using a shortcut, I push FUNC 6 to bring up the [Position [Tool Offset function and then push the 2 button. The number 2 tool's offset is now loaded and waiting to be used. I have 5 choices:

rear (push button 8)

left (push button 4) no offset (push button 5) right (push button 6)

front (push button 2)

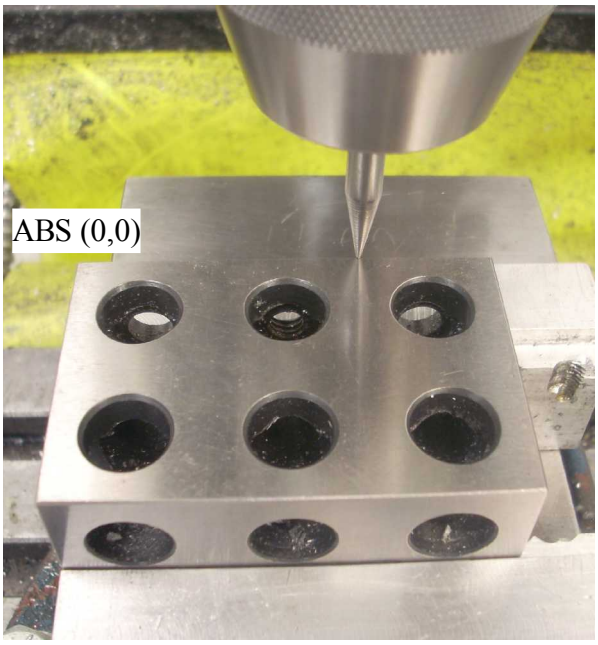


Since I will be cutting on the front side of the end mill, I push the 2 button. The end mill will barely scratch the back face of the block and the Y display will read 0.0000. If later I want to remove the offset, I just press the 5 button.

You can imagine milling a pocket and need to change offsets as you go around the hole. First you cut the left wall so press 4. Then you cut the back wall so before moving into position, press 8. Then you press 6 to cut the right wall. And finally, you press 2 to cut the front wall. Finish by pressing 5 so you remove the offset. Sure is handy!

[Position [Centerline

Plans sometimes specify a feature as being on the centerline of a part.



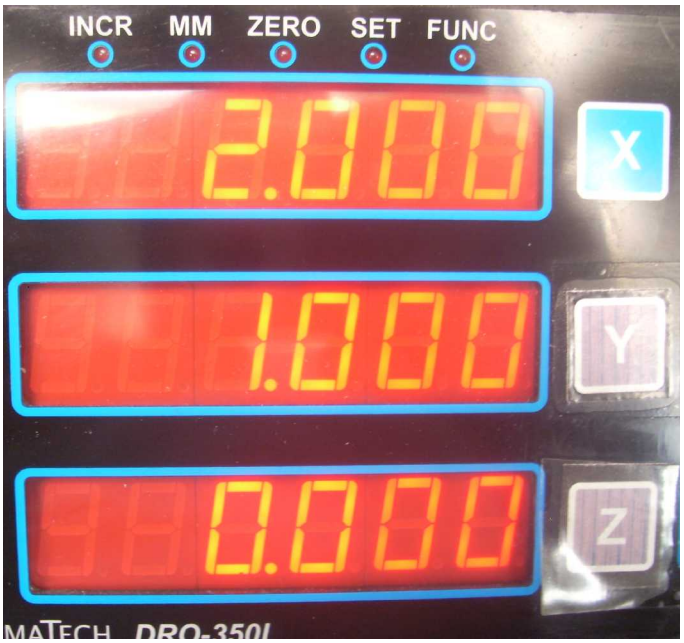
My ABS origin was previously set up to be the back left corner of the fixed jaw step. When I drop in my 1-2-3 block, its back left corner will be at the origin. I have moved my spud to X = 2.0000" and Y = 0.0000". The goal is to find the centerline of the block along the Y axis.

My Y axis crank is turned until my spud is lined up with the front face of the 1-2-3 block.

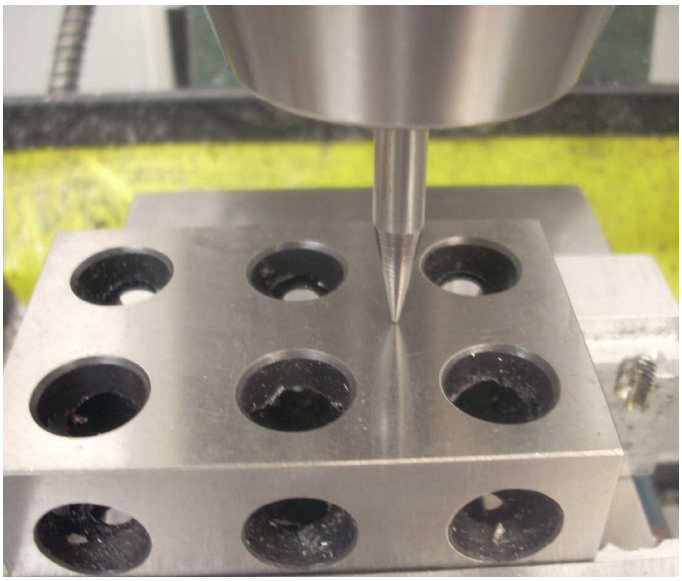
The Y display shows 2.0000" as expected since this is a 2" wide precision ground block.

The function is accessed by pressing FUNC, pushing the Z preset until you see [Position [Center and then press Enter. You can also used the shortcut FUNC followed by pressing "1". This tells the software you want to halve one of the displayed values in order to calculate the center line along that axis.

The software responds with this prompt of "CENTER SELECT" We want the Y axis so push the Y button.

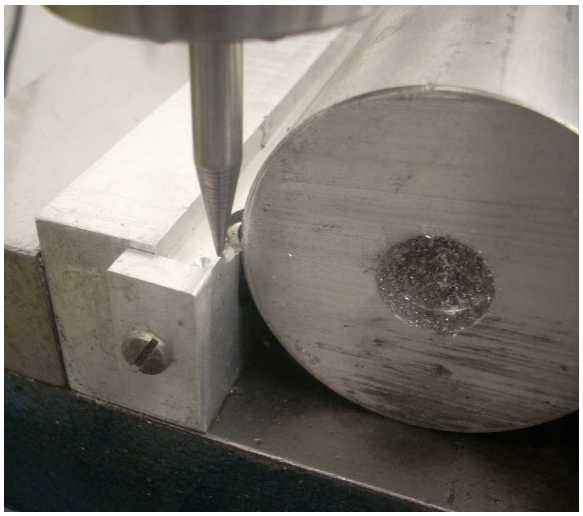


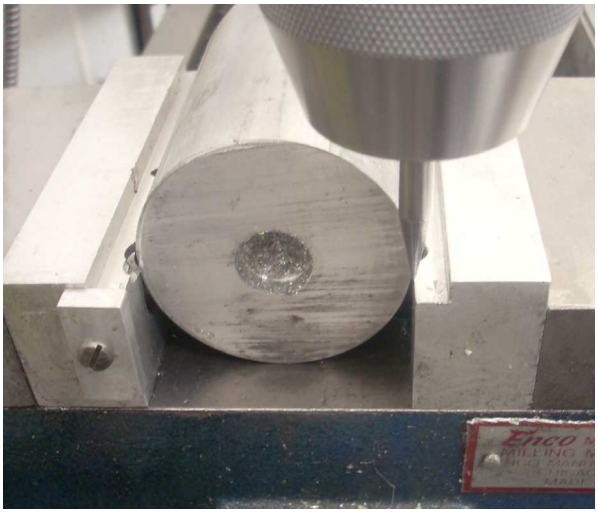
Our Y axis now shows a value that is half of the previous one. We can now crank the Y axis until we get to a Y = 0 which will be our centerline.



You can see that my spud is now at the centerline of the block along the Y axis.

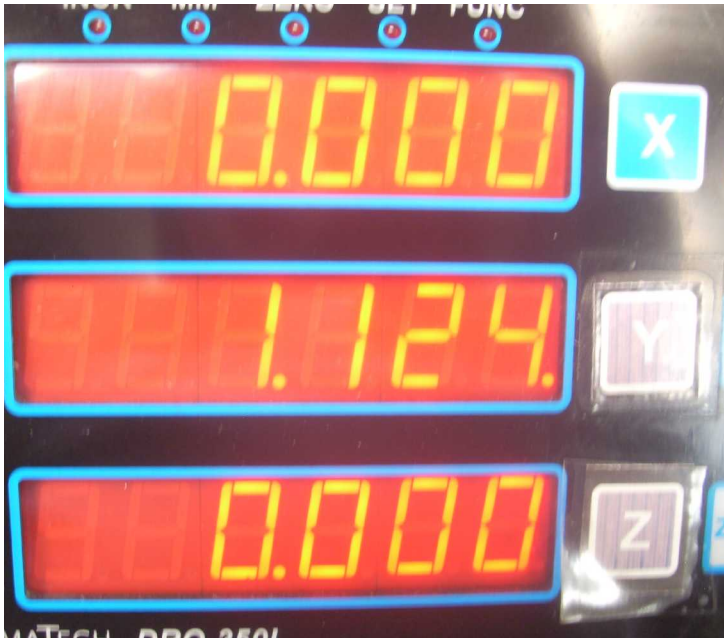
The Centerline Function works equally well when in INCR mode.





Another common need for finding a centerline is with round stock. In this case I must define a new value for Y = 0 because the round stock does not touch the vertical surface of the back lip. If I wanted to preserve my ABS Y = 0 value, I would use INCR here and define my INCR Y = 0 as shown. I then move the Y axis until I reach the front jaws face.

The round stock's diameter is 2.249”.



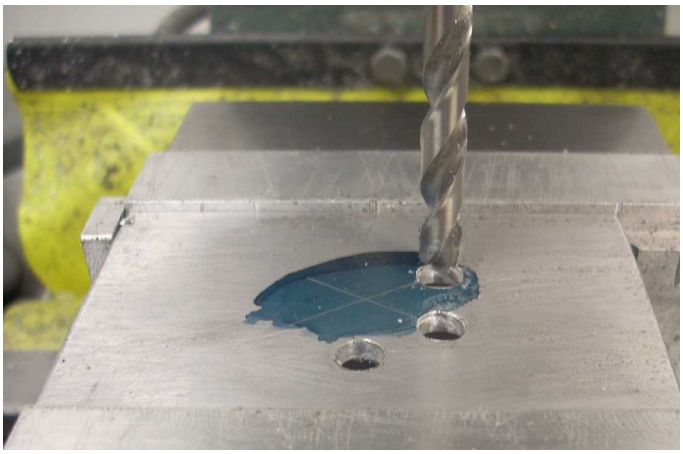
When I press FUNC and then “1” followed by pushing the Y preset, I get which tells me that my Y axis zero is now 1.1245” back from the present location.

I turned the round stock end for end so you can see how the spud lines up with its center.

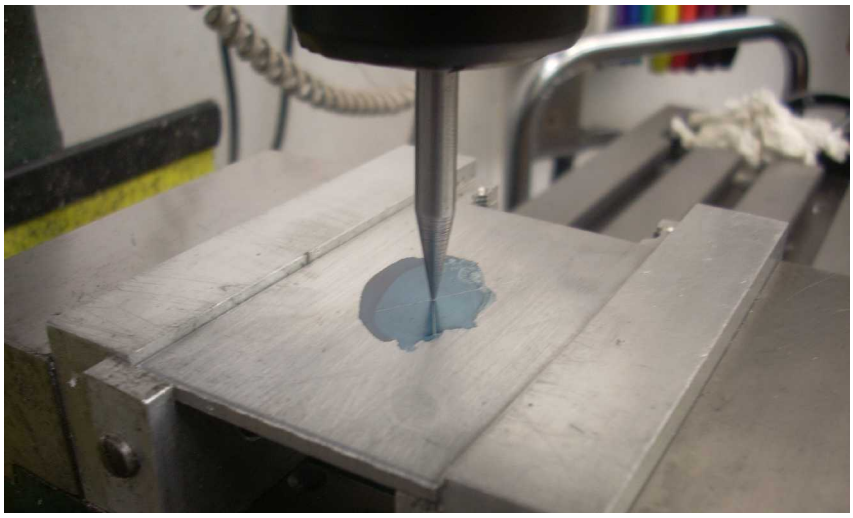
I have used a spud to illustrate the centerline function but it works equally well with an edge finder as Larry Gill has pointed out. You do not need to use the tool offset function either. Say you define the center of the edge finder as zero. You move to the fixed jaw and touch down. You are now a radius away from the fixed jaw. Next you move to the movable jaw and touch down. Again, you are a radius away. But since you have this same error on both ends of the line, the centerline is still in the right place. Similarly, you can use the edge finder against the part and get the same result. Do be careful to keep the height of the edge finder consistent on both faces of the part.

Now, if your vise jaws have V grooves in them, you should touch down only on the part. The jaws will be closer together than the diameter of the round stock.

[Define [Bolt Hole Circle Pattern and [Position [Bolt Hole Circle Pattern



There is more than one way to drill a series of holes around a circle. One common way is to use a rotary table. It is a simple matter to drill a hole, turn a predetermined number of degrees, and drill the next hole. Another method is to use the Bolt Hole circle Pattern function. Here you see a sample. I have drilled 3 holes on a 0.5" radius that starts at 0 degrees and is equally spaced around the circle for 180 degrees. Note that the pattern is hole-space-hole. This means that after the third hole there is a space. The end of that space is at 180 degrees.



Before I start drilling, it is necessary to move the spindle to the center of the bolt hole circle.

I then go to incremental mode and zero my X and Y axis. The function will use this incremental origin as the center of the bolt hole circle.

It is time to define the bolt hole circle pattern. I press FUNC, then push the Y preset until I see [define in the Y display. Then I press the Z preset until I see [bolt hole. Pressing Enter starts the function.

The first prompt is to name the pattern you are about to define. I pressed "1" and then Enter.

How many holes do you want on the circle?

I chose 3. Sure hope I never need to enter 99! It could be handy if I was chain drilling a large hole. In fact, if you used a 2 flute end mill instead of a drill, you could us this function to rough cut a disk without using a rotary table.

Pressing Enter records the number and moves us to the next prompt.

I now enter the radius of the circle. I chose a radius of 0.5”

We are being asked to define the starting angle for the circle. Because I have my X axis set to [reverse [no and my Y axis set to [reverse [yes, hole 1 will be in the front and sequence counterclockwise. My angle will start in the front and also go counter clockwise.

I want to start at 0 degrees so just press Enter.

And at what angle do you want the arc of holes to end? If you just press Enter, the holes will be equally spaced around the entire circle.

I chose 180 degrees and then pressed Enter. Given my choices on axis polarity, this angle is at the back of the table.

Bolt Hole Circle Pattern 1 has now been defined.

Now we will use this pattern. Pressing FUNC and then repeated pushes of the Z preset brings us to this prompt.

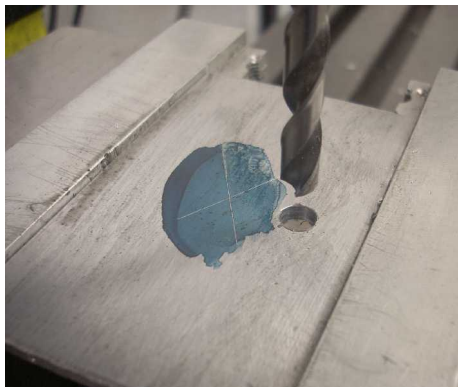
Press Enter to accept it and receive:

We answer this request for a defined bolt hole circle pattern by pressing “1” and then Enter.

Recall that our incremental origin was set at the center of our circle. The program now changes it to the center of our first bolt hole.

We are being told that the center of bolt hole 1 is zero inches along the X axis and -1.000 inches along the Y axis.

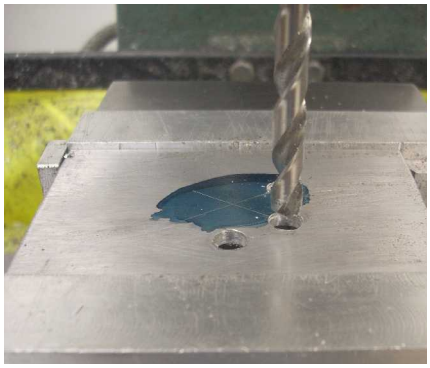
I turn my Y axis dial until the Y display reads 0.0000 inches. It was at -1.0000 so move in a positive direction to get to 0.0000. Given the definition of my Y axis, this means the cutter moves toward the front.



After drilling the hole at this new incremental origin, I press Enter to say I am done with hole 1.

OK, here comes bolt hole 2.

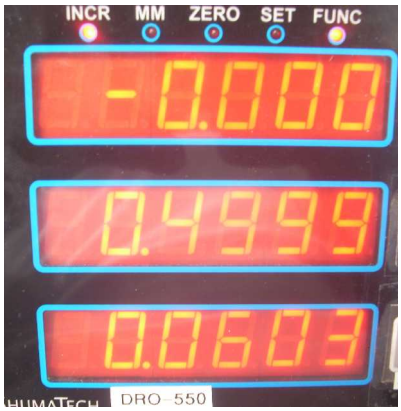
My incremental origin has now been moved to the center of hole 2. I must move my X axis +0.433 inches and my Y axis -0.2500 inches in order to reach it.

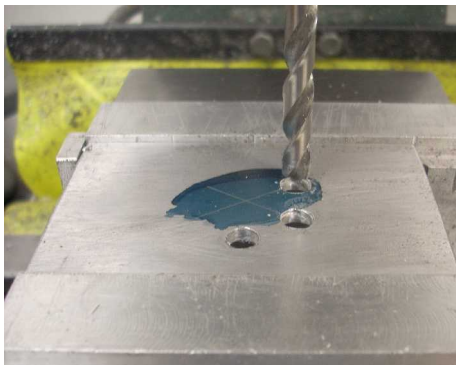


My second hole is then drilled. Enter is then pressed to advance to our third and final hole.

Pressing Enter again brings us the display of the final coordinates.

We must move the Y axis – 0.4999 inches in order to reach this last incremental origin.





After drilling our third hole, we press Enter.

That was the last hole. Press Enter to exit the function.

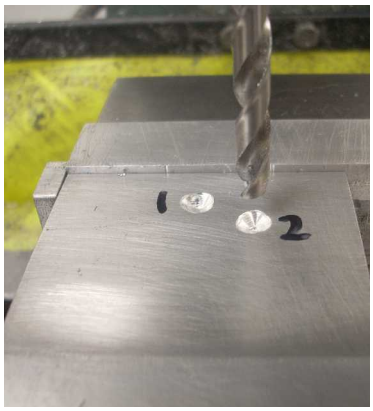
Note that each hole has associated with it the space to the next hole. So in my example I have 3 holes in 180 degrees and the 3rd hole is at 120 degrees. The end of its space is at 180 degrees. If you really want 3 holes with the last hole at 180 degrees, you should define 3 holes with a larger pattern angle. Hole 1 is at zero degrees, hole 2 is at 90 degrees, hole 3 is at 180 degrees. With each hole spaced 90 degrees from the last one, the pattern angle should be set to 180 + 90 = 270 degrees. That will put the end of the last hole's space at 270.

[Define [Grid Hole Pattern and [Position [Grid Hole Pattern

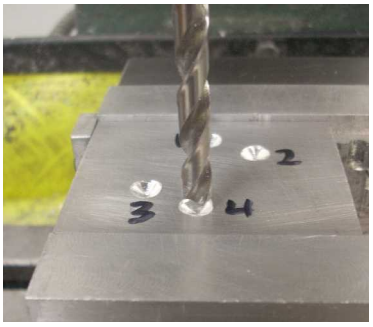
I have my X axis set to [reverse [no and my Y axis set to [reverse [yes. This means that my grid pattern is flipped from the default of X and Y [reverse [no. See the User's Guide for details.

Just for grins, my Grid pattern 2 has been defined as having an X distance of 0.5", a Y distance of 1.0", 2 holes along the X axis, and 2 holes along the Y axis. The angle is set to 30 degrees.

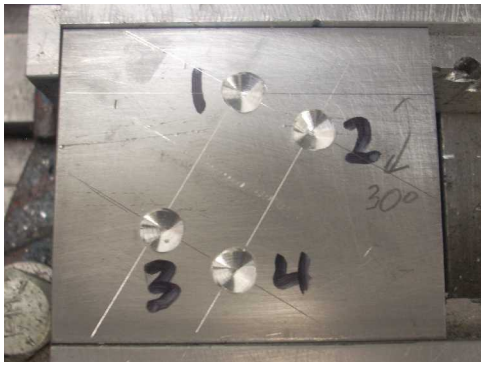
Here I have positioned my drill over my scrap of aluminum and zeroed my incremental X and Y axes.



I drilled down just enough to mark the location for hole 1. Enter was then pressed which told me I was about to drill hole 2. After cranking the X and Y dials until I read (0,0), I drilled my second hole.



Skipping ahead a bit, you can see that I drilled holes 3 and 4.



I have scribed lines between centers plus marked out the angle. Note that the 4 hole array has been rotated by 30 degrees. The holes are each 1/4" in diameter. The distance between holes 1 and 2 is 0.5". Between holes 1 and 3 is 1.0".

I'm not sure when I would need to rotate an array of holes, but certainly without any rotation, it will be a useful option.

These next 3 examples use Grid to chain drill a slot. I have defined Grid pattern 1 as

X dimension	0.75”
Y dimension	0
X holes	4
Y holes	1

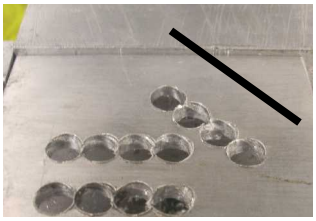
This means the distance between the center of the first hole and the last hole along the X axis is 0.75”. Since I am chain drilling a slot, there are no holes along the Y axis so I set the Y dimension to 0. I want 4 holes along the X axis. By setting the Y hole count to 1 I am saying there are no new holes along the Y axis.

My drill is 0.25” in diameter so this arrangement will give me no web between holes. This often causes the drill to move sideways into the previous hole as it drills. Lets see what happens.

I scribed the front left corner of the slot and set my absolute zero there. Then I moved to (0.125, -0.125) so the drill is located at the start of the slot. I then changed to incremental and set my (0,0). Drill that first hole and push Enter.

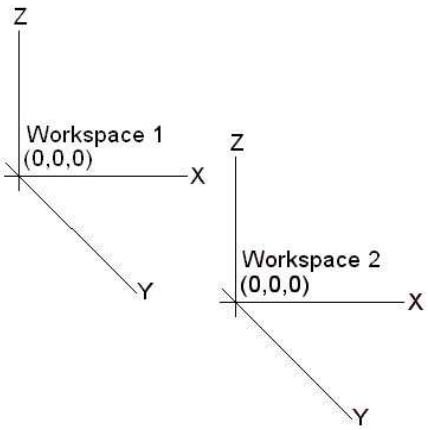
After drilling all 4 holes, you can see it came out fairly well. It doesn't take much time to get used to seeing a negative number show up on the X display and knowing that you must turn the crank in the positive direction to make it go to zero. Chain drilling goes very fast. Having the audible alert helps me to keep my eyes on the table motion until I get close to the next hole location.

I chain drilled a second slot with X dimension set to 0.765”. This gave me a web of 0.005” between holes.



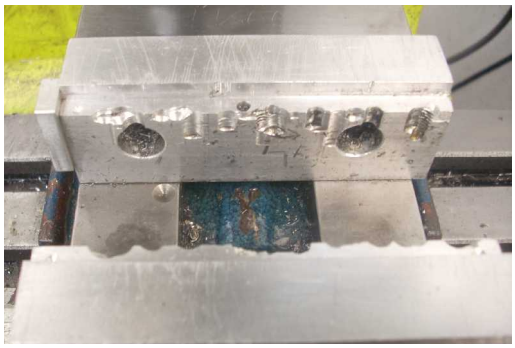
For my last example, I set X dimension to 0.77” and set the Grid angle to 45 degrees. Maybe some day I will need a slot at an arbitrary angle and don't want to use my rotary table.

[Position [Workspace



You can think of the DRO-550 as having multiple personalities. At the top level we can select which Machine we want to be. The two machines have nothing in common except one scale parameter. Within one of these machines, we have the ability to define up to 10 "Workspaces". Within each Workspace, we have our 3 absolute zero points and our 3 incremental zero points. This can be handy if you find yourself having to set up new zero points each time you clamp down a different part.

For example, say you set up work stops like this:



The step on the fixed jaw defines a Y position and the little block on the back left side defines a Z position. When I drop in part number 1, I press it firmly against these stops. I can define my absolute (0,0) point to be the back left corner.

My next part also contacts these stops but I want my absolute (0,0) point to be 1.0000" to the right and 0.4000" forward. Not a problem. I first move the table to the new origin. Then I change to Workspace 2 and set my absolute (0,0).

By switching between Workspace 1 and 2, I have access to two unrelated absolute (0,0) points.

Another potential application is having an array of bolt hole circle patterns on a single plate. Go to Workspace 1 and define the first incremental origin. The change to Workspace 2 and define the next incremental origin. You can do this up to 10 times. Then go back to Workspace 1 and call up the bolt hole circle pattern. When done drilling, change to the next workspace and repeat the same pattern.

Tachometer

[Setup [Machine followed by [Tach [Flex1 or 2 and [Status [Tachometer

The DRO-550 has two ports able to accept a tachometer. They are called "Flex" ports because the software can configure them to be inputs for tachometer signals or outputs to drive a piezo beeper. I have a tachometer built into my mill. It is of limited value for RPM because my spindle speed is set by a pair of belts. I read off the speed with each of the 12 possible belt positions and I'm done. Of more value is using the tachometer output in conjunction with the tool offset's tool diameter to calculate Surface Feet per Minute. Now, if I had a variable speed mill, the tachometer would be used during each machine set up.

More on the tachometer can be found at

http://shumatech.com/products/dro-350/add-ons.htm

[Status [Feedrate

I own a variable speed drive for my X axis. Being able to monitor the feedrate can be helpful on critical jobs.

[Status [Calculator

Over the years I have found that some people swear by Reverse Polish Notation calculators and the rest swear at them. I bought an HP-35 pocket calculator in 1972 and have loved RPN calculators ever since that day. You will either understand why or call me nuts. If you are curious about RPN calculators and wonder what the attraction is, I suggest you go to

http://en.wikipedia.org/wiki/Reverse_Polish_notation

or

http://www.hpmuseum.org/rpn.htm

Acknowledgment

Thanks to Larry Gill for finding my typos.

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

Appendix: Electronic Edge Finder

I use my EEF a lot and really enjoy it. The basic idea is that you press the ZERO button for whichever axis you want to zero. Then you crank the table until the EEF touches the part. This causes an electrical circuit to complete and tells the DRO to zero the display. 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. Usually two tries is enough to give a very accurate zero set.

Don't forget our old friend [Position [Tool Offset explained on page 16. We will be touching down on one of the sides 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 16 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 [Position [Tool Offset and pick tool number 16. Then I would press the “8” button to tell the DRO I will be using the rear face of the tool. I would then press ZERO on my Y axis and crank until the EEF touched my reference surface. This zeros the display. Normally I will set up my X = 0 point at the same time so just need to press “4” and touch down with the left face of the EEF. For more on my EEF, see

http://rick.sparber.org/Articles/eef/eef1.0.htm

and

http://rick.sparber.org/Articles/ea/ea.pdf

In the end, I took the commercial EEF and modified it to work with the circuit. My home made EEF probe was not repeatable enough for me.