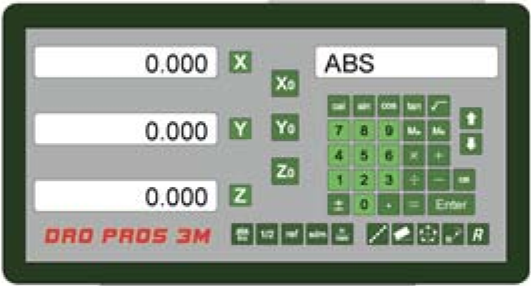
DRO PROS 3M



OPERATORS MANUAL

Thank you for purchasing a DRO PROS product. Please refer to our website, www.dropros.com for the most current updates and product information. We're proud of our products and we welcome your feedback! Please submit any errors, omissions or corrections you may see to dropros@aol.com. Your fellow machinists thank you!

DRO PROS 3M Milling Machine Operators Manual Version 10-3

DRO PROS 3M CHEAT SHEET

Set current position to zero

Switch display between inches and metric

Set Current Position to a set value

Finding the centerpoint of a workpiece

Position your tool against an edge · Reposition against opposite edge · Move table to 0.0000

Switch between ABS and INC mode:

Enter / exit calculator function

Result Transfer

Compute desired value · Transfer result to X axis · Move table to 0.0000

SDM Entry

Move table to desired point · Zero X and Y

Find REF

Arrow until - "FIND REF" · Push Axis to find · Move table until digits count

Recall 0

Arrow until - "RECALL 0" · Push Axis to find · Move table until digits count

Linear Hole Tool Position

Enter angle · Enter Distance · Enter Holes

Incline Tool Position

Enter angle · Zero the X Axis

Bolt Hole Circle

Enter Center coord's · Enter Diameter

Enter # Hole's · Start Angle · End Angle

Smooth ARC

Select Work Plane "R - XZ" · Center coord's

ARC radius · Start coord's

End Point · Tool Diameter

Tool Radius · R + TOOL

Simple ARC

Select Work Plane "S.R - XZ" · Select ARC Type

Select Radius · Select Tool Diameter

Parameters Setup

QUIT ➡ AXIS NO. ➡ DIRECTN ➡ RESOLU ➡ RAD/DIA ➡ LIN COMP ➡

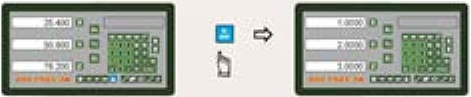
NL ERROR ➡ Z DIAL ➡ DIAL INC ➡ R MODE ➡ FLTR PR ➡ QUIT ➡

Thank you for purchasing a DRO PROS product. Please refer to our website, www.dropros.com for the most current updates and product information. We're proud of our products and we welcome your feedback! Please submit any errors, omissions or corrections you may see to dropros@aol.com
Your fellow machinists thank you!

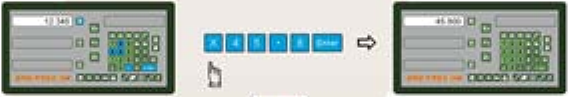
Set current position to zero

Switch display from inches to metric

Switch display from metric to inches

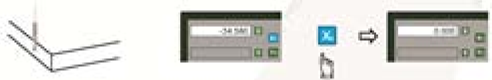


Set Current Position to a set value

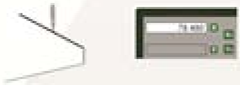


Finding the centerpoint of a workpiece

Step 1: Position your tool against one edge of the workpiece, then zero the display.

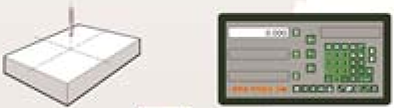


Step 2: Reposition your tool against the opposite edge of the workpiece.



Step 3: Push the appropriate axis button (X,Y or Z), then push the centerfind button.

Step 4: Move the table until "0.000" is displayed, indicating the centerpoint of the workpiece.



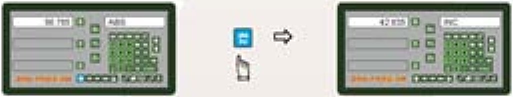
Absolute (ABS) versus Incremental (INC) coordinate systems

Two reference systems operate concurrently and independently of one another, the ABS (Absolute) reference system and the INC (Incremental) reference system. Zeroing either reference system does not zero out or alter the other systems zero reference. Both systems operate at all times; it matters not which coordinate system is selected or displayed. As the table is moved, both are continuously updated.

Typically Operators use the ABS coordinate system as the main reference system for their workpiece. Absolute zero (0,0) is typically set to a workpiece corner. Most all subsequent points and measurements are defined from this absolute reference point.

The Incremental or INC grid is most useful for determining incremental distances. Just like the ABS system, it can be referenced or zeroed at any time without affecting the other coordinate system. Typically this system is reserved for determining incremental distances from ABS points or coordinates.

EXAMPLE 1: Currently in ABS Mode, to switch to INC mode:



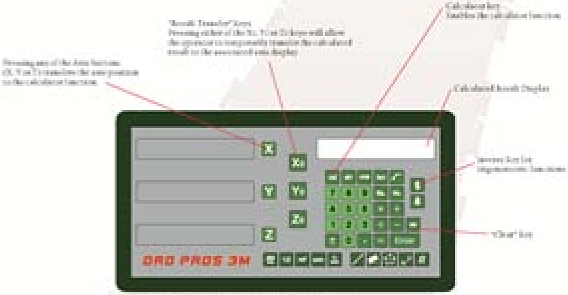
EXAMPLE 2: Currently in INC Mode, to switch to ABS mode:

Calculator Mode

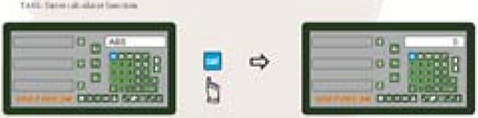
Calculator: It is often necessary to calculate values while machining. The built in calculator performs all normal functions: add, subtract, multiply and divide. Additionally, square root, sine, cosine, tangent and inverse functions are also provided.

The "result transfer" feature allows the calculated value to be temporarily transferred to any desired axis. This enables the table to be moved precisely to the calculated point.

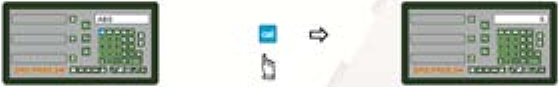
This feature saves time and reduces operator error as the resultant value need not be written down. Further, it eliminates the possibility of entering an incorrect value as the exact value is electronically transferred directly to the display head rather than manually entered.



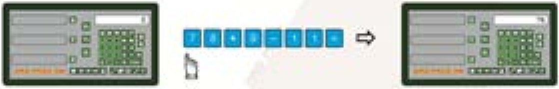
Key functions of the built in calculator



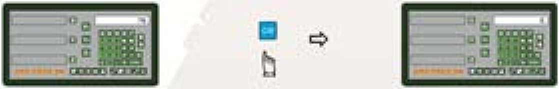
TASK: Enter calculator function



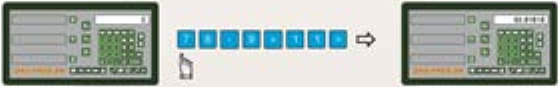
TASK: Perform a basic mathematical equation, ie [illegible]



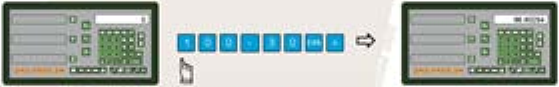
TASK: Clear the Calculator display



TASK: Perform a basic mathematical equation, ie [illegible]



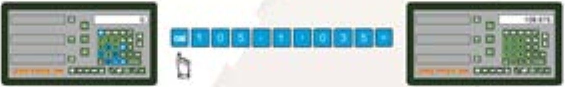
TASK: Perform trigonometric function, ie 100 x COS 30 = 86.602540

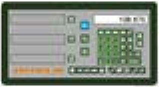


To calculate an inverse trigonometric function, i.e. $SIN^{-1}0.5 = 30°$

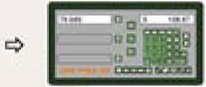
Result Transfer

Result Transfer allows the operator to move to a calculated or computed position. In this example, we're currently at 188.240mm on our X axis, but we want to move to a new position, in this case the product of 105 x 1.035, or 108.675. Of course, we could simply move the machine to 108.675, but there's always the chance we may transpose numbers and accidentally move to the wrong position. Result transfer eliminates the need to write down calculated results.



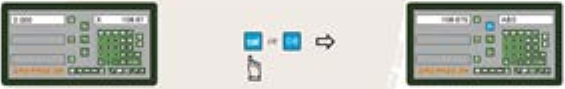


Push X₀ to transfer the calculated result (108.675) to the X axis



Note that the X axis window now displays the error or distance to go to get to our calculated point of 108.675. The original position 188.240 less the desired location of 108.675 = the distance to go to get to 108.675, or 79.565mm

Simply move the machine until the X axis displays "0.000", indicating we've reached our calculated position. To return back to the normal display, push the Calculator or Clear key. Note the X axis value of 108.675, our calculated value.



SDM (Sub Datum Memory) Feature

While having two separate datum coordinate systems is useful (ABS and INC), those wishing to machine repetitive tasks will find the Sub Datum Memory (SDM) feature to be extremely versatile and well suited to the task.

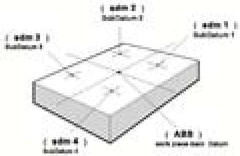
In short, the SDM function allows the operator to map out or remember 199 separate points on the workpiece. These points are referenced to the ABS zero point, and are unaffected by the INC zero point.

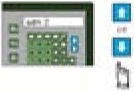
These points may be input to the system in two distinct methods. The first method, **Direct Entry**, employs the operator to physically move the machine to each individual point, at which time the operator pauses and enters the position into SDM memory. The second method, **List Entry**, allows the operator to enter the coordinates in a list form, without having to physically move the machine to each individual point. Both methods accomplish the exact same end result, that is, programming points into SDM memory.

Method 1: **Direct Entry** - Move the machine to each individual position.
Method 2: **List Entry** - Enter the coordinates in a list-like format without ever moving the machine.

Remember, all SDM waypoints reference the ABS coordinate system zero point. In other words, changing the ABS zero point will shift the SDM waypoints by the same increment on your workpiece.

All SDM points reference the ABS zero point.





To switch between SDM points, simply arrow up or down as needed.

Simply press the arrow keys to the appropriately numbered point you're searching for. The value displayed in the X and Y windows shows the error or distance to the selected point. Simply move the table so that "0.000" is displayed in each X and Y window.

NOTE: Do not zero out the display by pressing the Xo or Yo keys. Physically move or reposition the table so that the X and Y displays both indicate "0.000". This indicates you've reached the point.

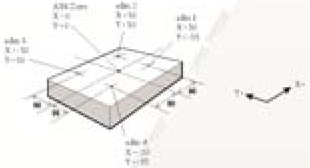
Press the up or down keys until the desired SDM point is reached. Then move the table until "0.000" is displayed in both X and Y windows.

Application Example:

As mentioned before, two methods can be used to enter coordinates into SDM memory:

Method 1: **Direct Entry** - Move the machine to each individual position

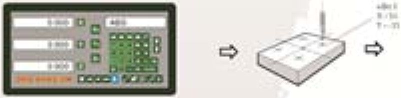
Method 2: **List Entry** - Enter the coordinates in a list-like format without ever moving the machine



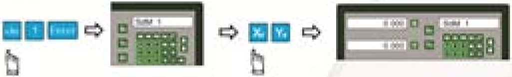
Direct Entry

Step 1: First, select the ABS coordinate system. Next, position your workpiece on your table and set the appropriate ABS zero point. Remember, all SDM points will reference whatever ABS zero point you set. In the example below, we've chosen to set the centerpoint of the project as our ABS zero.

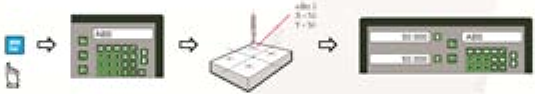
Step 2: We've decided to locate SDM point 1 at X-50, Y - 35 from ABS zero. Move the machine to these coordinates.



Step 3: Switch to SDM mode Point 1, then zero the coordinates to set current position as SDM Point 1



Step 4: Switch back to ABS mode, then move the machine to SDM point 2 (X = 50, Y = 50)

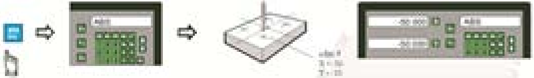


Step 5: Switch to SDM mode Point 2, then zero the coordinates to set current position as SDM Point 2

Step 6: Switch back to ABS mode, then move the machine to SDM point 3 (X = -50, Y = 50)

Step 7: Switch to SDM mode Point 3, then zero the coordinates to set current position as SDM Point 3

Step 8: Switch back to ABS mode, then move the machine to SPM point 4 (X = -50, Y = 15)



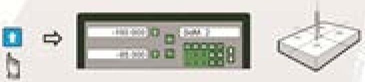
Step 9: Switch to SPM mode Point 4, then zero the coordinates to set current position as SPM Point 4

At this point, all 4 SPM points have been set up!

In order to switch to SPM point 1, simply arrow down three times to select SPM point 1.

Note that the machine is physically still located at SPM Point 4. The display window is simply indicating the error between the present position and selected SPM point 1. In other words, to get to SPM Point 1, we need to travel 100mm to the right.

We can switch to SPM waypoint 2 by pressing the up arrow.



From the current position, SPM 2's relative position is 100 to the right and 45 to the front of the workpiece. If we were to move the table so that the display read "0.000" in both axes, we would be directly overtop of SPM point 2.

We can now move to SDM waypoint 1 by pressing the down arrow

To switch back to the ABS coordinate system, simply push the ABS/INC button.

Note how much the display values changed. The display is now referencing the ABS zero point, and is telling us relative to the ABS zero we chose (in this case the centerpoint of the workpiece), we are 50mm to the left and 35mm to the rear of ABS zero.

List Entry

Method 2: **List Entry** (Entering waypoints in a list like format)

In a case where many SDM points need be input, the operator will most likely find the second method, or List Entry, to be a more expeditious and convenient method than Direct Entry. This is because the List Entry method does not require the machine to be physically moved to each individual point.

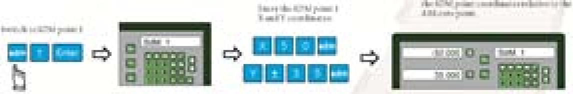
The List Entry method consists of entering SDM waypoint coordinates while physically remaining at the absolute zero point. All of the waypoints are entered in a list like format, without the need of physically moving the machine to each waypoint. In this respect, it is a much quicker method of coordinate entry than the Direct Entry method, which requires the machine to be physically moved to each point.

Step 1: Ensure the display is in the ABS coordinate system.

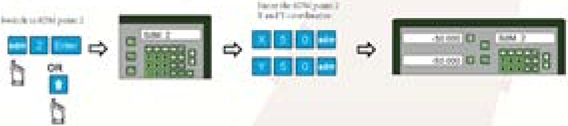
Step 2: Establish and locate the tool at your workpiece centerpoint

Step 3: Establish your ABS zero point by zeroing out the X and Y axis

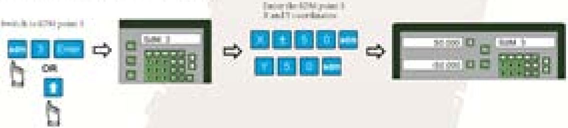
Step 4: Enter the coordinates for SDM point 1



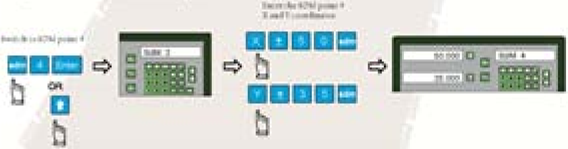
Step 5: Enter the coordinates for SDM point 2



Step 6: Enter the coordinates for SDM point 3



Step 7: Enter the coordinates for SDM point 4

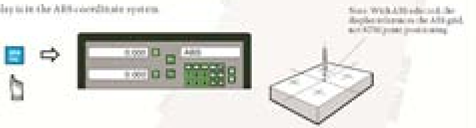


At this point, all 4 SDM points have been established

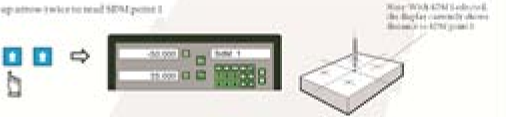
Simply scroll up or down to read the desired SDM point

Recalling SDM Points

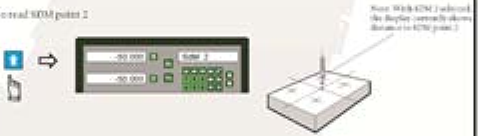
Ensure the display is in the ABS coordinate system



Push the up arrow twice to read SDM point 1



Scroll up again to read SDM point 2



Scroll down to return to SDM point 1

Power Off Memory

Power Off Memory is a function that helps restore the workpiece zero point. Losing the reference zero point can happen if the machine is moved with the DRO unit powered off. Fortunately, the "Power Off Memory" function allows the exact workpiece zero point to be 'recaptured' very easily.

Every glass grating scale has a mid or reference point. When the Operator establishes a new Absolute or Incremental zero point, the DRO establishes a snapshot of the desired zero point versus the scales actual midpoint. While this is transparent to the Operator, the important thing is that this offset is permanently stored in the DRO's memory until a new zero point is set.

If for some reason the zero point is lost, as in the case when a DRO is moved without power, the exact zero point can still be recovered. All we need to do is simply recover the zero position by recapturing the stored distance from the ref point.

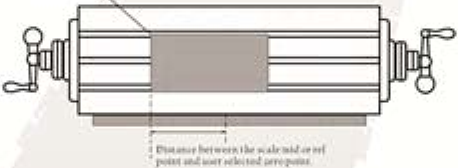
IMPORTANT NOTE: When a new ABS zero point has been established, the reader head must travel across the mid or ref point of the scale. This enables the DRO to 'snapshot' the difference between the user selected ABS and the ref point, and enables the ref datum recall function.

The Power Off Memory feature consists of two components:

Find Ref Function
Recall 0 Function

The **Find Ref Function** normally ensures the ref point or mid point of the scale has been captured. It ensures after inserting a new ABS zero point that the mid point on the scale has indeed been found and the **Recall 0 Function** will be available if needed.

The **Recall 0 Function** describes the steps necessary to recapture a lost zero point.



In other words, the **Find Ref Function** is accomplished <u>before</u> a power failure as a preventive measure to ensure the user selected zero point can be re-established.

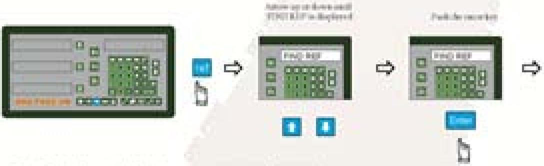
The **Recall 0 Function** serves to re-establish a lost zero point <u>after</u> a power failure or if the mill is accidentally moved without power.

Any time a new zero point is stored, whether by setting a new ABS or INC zero, coordinate preset, or centre find function, the DRO will automatically store the relative distance between ABS and the scales ref point. As long as at some point in time the reader head travels across the mid or ref point of the scale, the Power Off Memory Function is enabled. Whether the Operator crosses the mid point of the scale accidentally through the course of the work session or purposefully through the use of the 'Find Ref' function matters not, both "methods" enable the DRO to "snapshot" the difference between the user selected ABS and the ref point, thereby enabling the ref datum memory function.

Find Ref Function

TASK: Use **FIND REF** function to manually capture a scales ref point in case Power Off Memory is overextended.

Step 1: Push the 'ref' function key on the front of the display



Step 2: Select the axis which you're trying to recapture the zero point

NOTE: Selecting 'ref' selects all 3 axis

Step 3: Move the machine across the centre, or mid point of the scale, until the digits start counting again

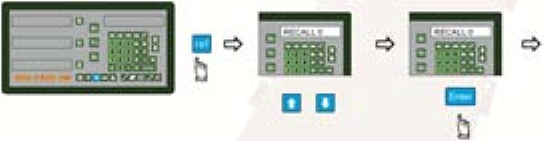
Recall 0 Function

TASK: Use 'Recall 0' function to restore a scales lost zero point in case of power failure or operator error.

Step 1: Push the 'ref' function key on the front of the display

Arrow up or down until RECALL 0 is displayed

Push the enter key



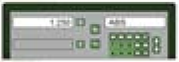
Step 2: Select the axis which you're trying to restore the zero point

In this example, press X to restore the X axis

Note: "X REF" starts flashing until the mid point is captured

Step 3: Move the machine across the center, or mid point of the scale, until the X axis digits start "counting"

Move the machine such that the reader head crosses the mid point of the scale

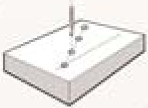


As soon as numbers appear on the display, the zero point has been restored

Linear Hole Tool Positioning

The Line Hole function allows the operator to drill a series of holes in a row or line. The operator can specify the desired angle of placement, the spacing or distance, and the required number of holes.

Simply enter the Line Angle (LIN ANG), Line Distance (LIN DIST) and number of holes (NO. HOLE) and the DRO calculates all hole coordinates automatically. Simply arrow up or down to the desired number hole, then move the machine until the display reads "0.000" on the X and Y axis.



For subsequent holes, simply arrow up or down as required.

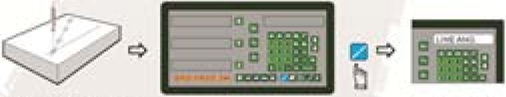
- to proceed to the next hole
- to proceed to the previous hole

Angular Direction:
Clockwise - Negative angle
Counter clockwise - Positive angle

Linear Hole Tool Positioning - Setup

Line Angle	LINE ANG.	10 degrees (counter clockwise)
Line Distance	LIN. DIST.	90.000mm
Number Holes	NO. HOLE	4

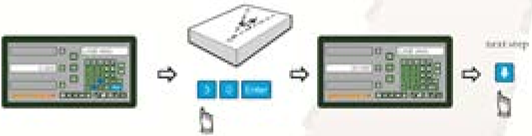
The Line Hole function allows the operator to set a given number of holes in a straight line. The function allows the operator to choose the number of points, the distance or spread between the first and last hole, and the angle upon which the points are set. The function uses the present physical position as the starting point, so it is important to pre-position the workpiece to the location of the first hole before the Line Hole function is started.



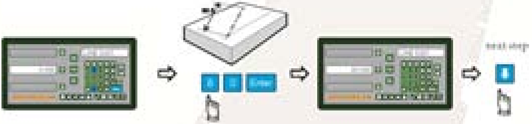
Step 1: First, locate the tool at the first hole position

Second, push the Line Hole function button

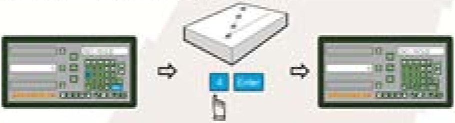
Step 2: Enter the desired Line Angle - (In this case, 30 degrees)



Step 3: Enter the desired Line Distance - (In this case, 80 millimeters)



Step 4: Enter the desired Number of Holes - (In this case, 4 Holes)

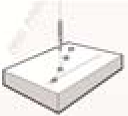


Linear Hole Tool Positioning - Example

At this point, all required Line Hole parameters have been entered. To proceed to the first hole, simply press the down arrow key.

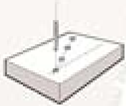
Since the Line Hole function was started at the location of the first hole, both the X and Y axis display will read "0.000". As soon as you step or arrow to the following holes, the display will read the error or distance to go to the following holes. Simply move your machine until the display reads "0.000", indicating you are now precisely "at" the location of the hole.

Press the down arrow key to get to Line Hole #2.



Move the machine so the display =0.000

Press the up arrow key to return to Line Hole #1.



Move the machine so the display =0.000

To temporarily return to the ABS coordinate system, press the "0" key

Presently in Line Hole function

Temporarily in ABS coordinate system

To return to the Line Hole function, press the "0" key

Temporarily in ABS coordinate system

Back in Line Hole function

To exit Line Hole function and permanently return to the ABS coordinate system, press the Line Hole function key

Presently in Line Hole function

Permanently back in ABS coordinate system

Incline Tool Position

It is sometimes necessary to tilt the workpiece at an angle relative to the travel of the machine. To ensure the workpiece is properly cutted at the desired angle, the Operator can use the Incline Tool Position function.

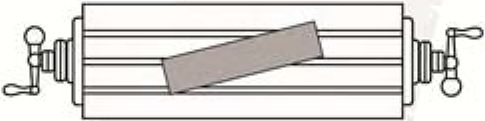
It is important to note that the angle of the workpiece is referenced to the tables X axis. In the following example, a 20 degree counter-clockwise tilt (i.e. 20 degrees) is selected as the angle.

Angular Direction:
Clockwise - Negative angle
Counter clockwise - Positive angle

Incline Tool Position - Example

In the following example, the workpiece is angled at a 20 degree angle to the worktable, ie it is angled 20 degrees counterclockwise from the tables back and forth X axis movement.

Step 1: Install the workpiece onto the work table at approximately a 20 degree counter-clockwise angle.

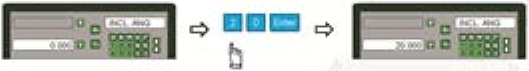


Step 2: Push the Incline Tool position function button.

Pressing the up or down arrow keys will cycle through the available options, ie XY, XZ and YZ.

When finished selecting, push the enter key to continue to the next parameter.

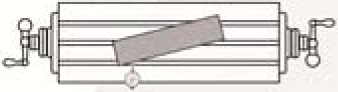
Step 3: Enter the Incline Tool Position desired angle (20 degrees).



Step 4: Press the down arrow key

At this point, all required parameters have been entered. The Incline Tool position feature is ready to use!

Now, zero your dial indicator on one end of the workpiece:



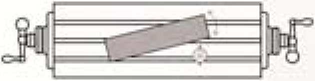
Next, zero the X axis readout

Notice how zeroing the X axis also zeroed out the Y axis display

Now that your in INCL mode, the Y display will show the error or distance from your 20 degree angle. Since you zeroed the X axis reading, the Y axis display also zeroed (Because it is assumed at this point to be "on the line"). As you subsequently move along the length of the workpiece, the Y axis display will show error or distance from the 20 degree line. If the workpiece happened to be perfectly aligned, as you move along the X axis, and subsequently move the machines Y axis to read 0.000, the dial indicator would also read 0. Most likely, though, you'll need to adjust the alignment of the workpiece until the dial indicator also reads zero. Let's take a look at an example...

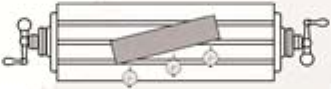
After moving along the x axis an arbitrary amount to the other side of the workpiece (in this case 123.55), we move the machines Y axis until 0.000 is displayed on our digital readout.

Note that the dial indicator does not read zero, which means the workpiece is not on a "true" 20 degree line. To correct the workpiece to the desired angle, the right side of the workpiece needs to be moved or rotated by the amount shown on the dial indicator in order for the workpiece to be put back on a 20 degree line.



When repositioning the workpiece it is important to remember that any angular adjustment on one end will likely affect the position on the other end. Therefore, this procedure of zeroing the dial indicator on one end and then checking the error on the other end will need to be carried out several times in order to obtain a satisfactory alignment.

In the illustration below, the right side of the workpiece has been rotated to bring it in alignment, and no matter where the dial indicator is placed along the workpiece (assuming the Y axis is moved to indicate 0.0000), the dial indicator will read 0 also.



To temporarily return to the ABS coordinate system, press the "0" key.

Presently in Incline Tool function | Temporarily in ABS coordinate system

To return to the Incline Tool function, press the "0" key.

Temporarily in ABS coordinate system | Back in Incline Tool function

Bolt Hole Circle

The Bolt Hole Circle function allows the operator to set a desired number of holes in a circular pattern. While most projects will most likely consist of holes evenly spaced around a complete circle, Bolt Hole Circle further allows the operator to designate a starting and stopping angle, such that if the holes were desired to be evenly spaced only along an arc of only 60 degrees then this too would be quite possible. Aside from a less than complete circle, choosing the starting and stopping angles allows the operator to very specifically designate at which angle the "bolt hole" pattern starts and ends.

The parameters required to be entered for Bolt Hole Circle are the following:

Centre	CENTER
Diameter	DIA
Number of Holes	NO. HOLE
Starting Angle	ST. ANGLE
Ending Angle	END. ANG

Angular Direction:
Clockwise - Negative angle
Counter clockwise - Positive angle

Bolt Hole Circle - Setup

Center coordinates (CENTRE)	X = 0.000
Diameter (DIA)	80.000
Number Holes (NO HOLE)	5
Starting angle (ST ANG)	30 degrees
End angle (END ANG)	60 degrees

Step 1: Locate your hole center and press the Bolt Hole Circle function key

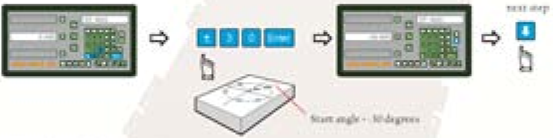
Press the Bolt Hole Circle function key

Step 2: Enter the Centre coordinates

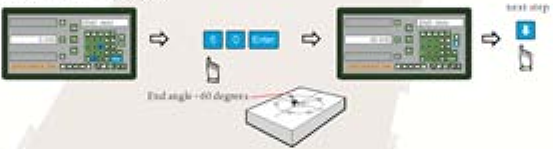
Step 3: Enter the Diameter (DIA)

Step 4: Enter the number of holes (NO.HOLE)

Step 5: Enter the start angle (ST.ANG.)



Step 6: Enter the end angle (END ANG.)



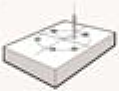
At this point, all parameters for the Bolt Hole Circle function have been programmed.

Bolt Hole Circle - Example

The Operator can now [up key] or [down key] to select which hole to move to, then simply move the machine to zero the display, which means the target hole has been reached.

To move to the next hole

Move the machine to zero the display



Once the display has zeroed, this means target hole #2 is centered

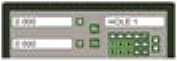
To move to the previous hole

Move the machine to zero the display

Once the display has zeroed, this means target hole #1 is centered

To temporarily return to the ABS coordinate system, press the "0" key

Presently in Bolt Hole Circle function



Temporarily in ABS coordinate system

To return to the Bolt Hole Circle function, press the "0" key

Back in Bolt Hole Circle function

To exit Bolt Hole Circle function and permanently return to the ABS coordinate system, press the Bolt Hole Circle function key

Presently in Bolt Hole Circle function

Permanently back in ABS coordinate system

ARC Machining

ARC machining allows the operator to machine a round corner or arc. While this can be a fairly time consuming process and would not be recommended where a great number of corners must be cut on a regular basis, for the occasional arc it is an extremely useful feature to have.

It is important to note that an arc or corner can be cut in any axis - XY, YZ or XZ. Additionally, it should be noted that both 2 and 3 axis displays are capable of performing any of these arcs. Although the 2 axis display does not have a third or Z axis, the display can simulate the third axis through the use of the Z Dial, Dial Inc and R Mode parameters found in the setup menu.

There are two ARC functions from which to choose from:

Smooth ARC - Takes a little longer but yields a finer finish
Simple ARC - easier to setup and quicker to cut

Smooth ARC vs. Simple ARC

The Smooth ARC function provides the maximum flexibility in ARC machining as all arc parameters are individually defined and not "preset" as in Simple ARC. Specifically, the operator must program the following:

Arc Centerpoint
Arc Radius
Arc Startpoint
Arc Endpoint

Although more time consuming, the advantage of "Smooth ARC" is that the Operator defines all the variables, thereby making "Smooth ARC" a more powerful function than "Simple ARC".

"Smooth ARC" can also machine virtually every type of ARC, even intersecting ARC's.

The drawback to "ARC" is twofold - first, the operator must take the time to input the parameters, and second, in general it takes longer to cut as there are typically many more cutting steps or passes involved.

Smooth ARC

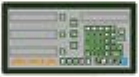
Smooth ARC provides the maximum capability in arc machining as it allows the user to define all arc parameters. Specifically, the user defines:

ARC center point
Arc radius
Arc start point
Arc end point

Advantages: Smooth ARC function is extremely flexible, and can machine virtually all kinds of arcs, even intersecting arcs.

Limitations: Setup is more complicated as the user must designate all arc parameters.

Simple ARC



Simple ARC is primarily designed to machine simple arc or radius corners, and provides the user with quick, pre-configured arc parameters.

Advantages: Extremely simple to use, user need not even calculate arc parameters as they are already pre-programmed. As soon as the user positions the tool at the start point, arc machining can begin right away.

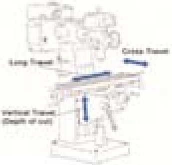
Limitations: User is restricted to eight preset arcs, and cannot machine more complicated or intersecting arcs.

When using the arc function, it is necessary to define the position of the arc center, arc start and end points. These points are all defined by coordinates.

Coordinates are a set of numbers representing a position on the workpiece. Typically, coordinates are defined by a point's relative position on three axis - specifically, the X, Y and Z axis.

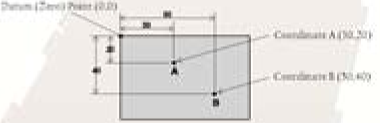
During scale installation, it is typical to set the scale read direction to the same direction as the machine. For a typical milling machine, read direction is typically matched to the lead screw calibration, which is as follows.

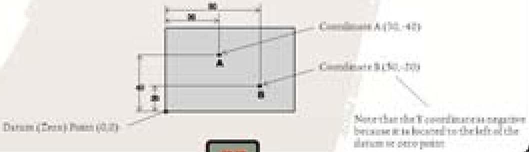
Understanding Coordinate Systems



Coordinate Examples

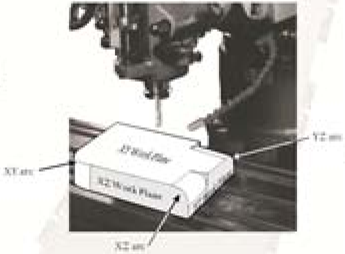
Whichever plane (XY, XZ or YZ) is chosen, coordinates designate a point's position relative to the datum, or (0,0) point. While a typical zero point is located on the workpiece, such as at the center point or left front edge, it truly is arbitrary, and in fact need not be located on the part itself. Further, the coordinates may be either positive or negative, depending on which relative 'side' or direction from the zero point they reside.





Work Planes

It is important to understand the three different work planes utilized by the ARC function. All three planes - XY, XZ and YZ are available in both two and three axis readouts. One of these planes must be selected in the 'arc setup' mode prior to machining.

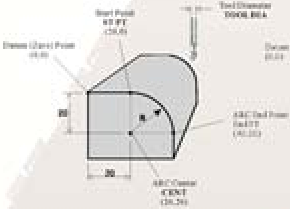


R+TOOL / R-TOOL

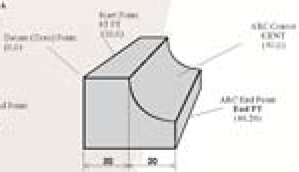
R+TOOL is used for machining 'outside' arcs, while R-TOOL is used for machining 'inside' arcs or edges.

	R+TOOL	R-TOOL
XZ / YZ Plane		
XY Plane		

R+TOOL
(Outside Radius ARC)

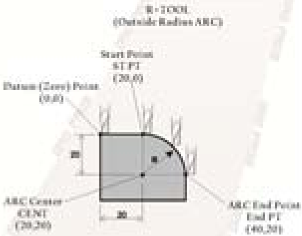


R-TOOL
(Inside Radius ARC)



For our example, the following parameters need to be entered for ARC machining:

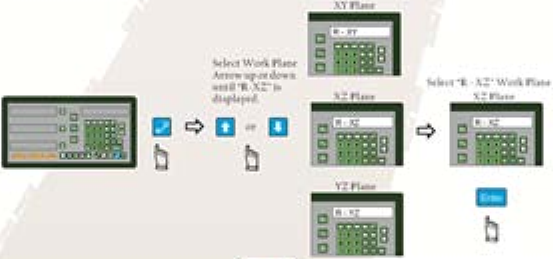
1. Work plane - XZ
2. Arc Center - CENT - (20,20)
3. Arc Radius - R - 20
4. Arc Start Point - ST PT - (20,0)
5. Arc End Point - End PT - (40,20)
6. Tool Diameter - TOOL DIA - 0mm
Why 0mm? Because we're using a flat end mill
7. Tool radius compensation - R+TOOL



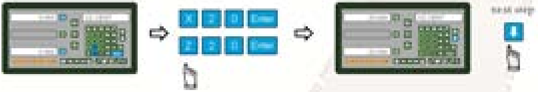
Smooth ARC Machining - Example

Step 1: Position the tool at the start point of the ARC. Note how the left edge of the end mill is aligned with the 0,0 point. That's because the edge of the mill will be doing the cutting - we're using a flat end mill, therefore we also set the tool diameter to 0.

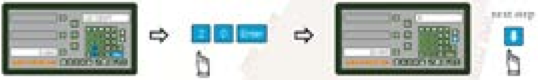
Step 2: Push the Smooth ARC function key, then select the work plane (in this case, XZ)



Step 3: Enter the ARC center coordinates (XZ CENTR) X-20, Z-20

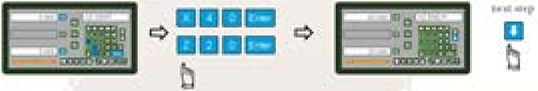


Step 4: Enter the ARC radius (R) - 20mm

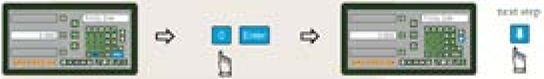


Step 5: Enter the start point coordinates (XZ ST PT) - X-20, Z-0

Step 6: Enter the end point coordinates (XZ END P) X-40, Z-20

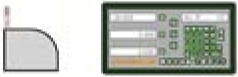


Step 7: Enter the tool diameter (TOOL DIA) 0mm

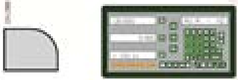


Step 8: Select tool radius compensation (R + Tool)

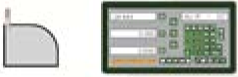
At this point all parameters have been entered into the display. The display calculates the arc parameters and shows the error or distance you are from the calculated arc. Initially, the display shows "0.000" on the X and Z axis as the cutting bit is "on the arc". Your first action should be to lower the cutting bit down "into the arc". As you do, the X axis display window will show the error or X axis distance from the arc. Simply reposition the table to zero out the error, make your cut, and then continue to repeat the process until the arc is completed.



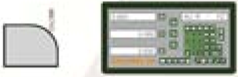
This is the starting position of our display. At this point, start the arc by moving the end mill down, into the arc.



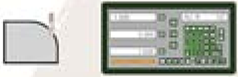
In this illustration the operator has accidentally moved the Z axis in the wrong direction, away from the arc. Accordingly, the Z axis window displays "Z OUT LF", meaning Z Out of Limits". Simply reposition the Z axis down, or "into the arc".



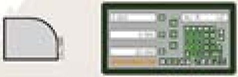
At this point the operator has moved the cutter down into the arc .5mm. Notice the X axis error of -24.44mm. Before making a cut, reposition the table to zero out this error. (Note the operator has chosen to perform the arc with .5mm Z steps or increments. This choice is purely arbitrary, you may choose as small or large as time allows.)



Now the operator is ready to make the first cut. Note the X axis display window indicates an error of 0.000, meaning it's "on the arc". The Z axis value of .5mm simply means the cutting tool is currently .5mm deep into our 20mm arc.



After the first cut, the operator stepped the Z axis down another .5mm for our second cut. Note total depth is now at 1mm. Make sure to zero out the X axis error of -1.805 before making the second cut!



This is what the display should look like after the last cut has been made. Note the Z axis depth is 20mm, and the cutting tool is still "on the arc" because the X axis window shows an error of "0.000".

Simple ARC Machining

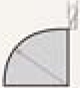
Simple ARC Demonstration

Although Simple Arc is not as powerful or customizable as Smooth ARC, most customers will find that Simple Arc is more than adequate for most arc operations. Simple ARC allows the user to choose from eight different preset arcs.

After deciding which preset arc to machine, all that remains is for the user to input the desired radius, tool compensation and machining increment or step.

Simple ARC Machining - Example

Step 1: For this example we will be machining a 20mm arc sweeping down to the left. Because the function is Simple Arc, it is necessary to position the tool at the start point of the ARC.



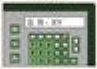
Radius - 20mm

Step 2: Enter the Simple Arc function

Step 3: Select the work plane (in this case, XZ)

Select Work Plane
Arrow up or down until "S. R. - XZ" is displayed.

XY Plane



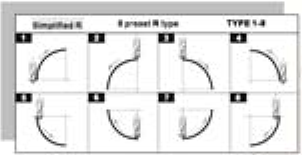
XZ Plane

YZ Plane

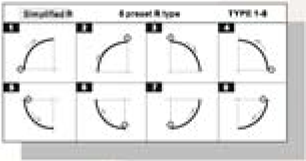
Select "S. R. - XZ" Work Plane
XZ Plane

Step 4: Select the preset arc type

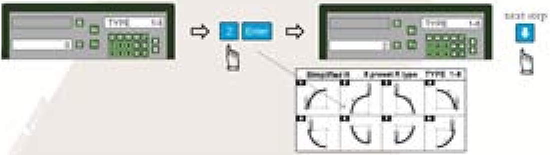
ARC types 1-8 for beveling an edge (XZ or YZ Plane)



ARC types 1-8 for rounding a corner (XY Plane)



In our example we're cutting a downward sweeping bevel cut - therefore we previously chose XZ as our work plane and we'll now choose "Type 2" as our type of cut.

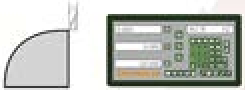


Step 5: Select the radius

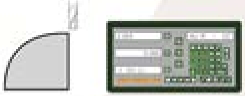
Step 6: Enter the tool diameter

Since we are using a flat end mill to machine an arc, the cutting edge is always the corner. Therefore, TOOL DIA must be set to 0.000.

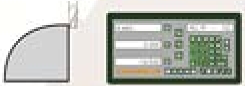
At this point, all parameters for Simple ARC machining have been programmed. The display should look similar to this



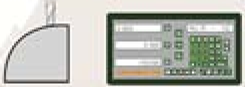
This is the starting position for Simple ARC. At this point, start the arc by moving the end mill down, into the arc.



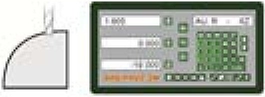
In this illustration the operator has accidentally moved the Z axis in the wrong direction, away from the arc. Accordingly, the Z axis window displays "Z OU LF", meaning Z Out of Limits". Simply reposition the Z axis down, or "into the arc".



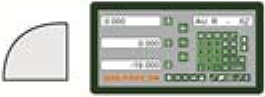
At this point the operator has moved the cutter down into the arc .5mm. Notice the X axis error of 4.44mm. Before making a cut, reposition the table to zero out this error. (Note the operator has chosen to perform the arc with .5mm Z steps or increments. This choice is purely arbitrary, you may choose as small or large as time allows.)



Now the operator has repositioned the table to zero out the X axis error and is ready to make the first cut. Note the X axis display window indicates an error of 0.000, meaning it's "on the arc". The Z axis value of .5mm simply means the cutting tool is currently .5mm deep into our 20mm arc.



After the first cut, the operator stepped the Z axis down another .5mm for our second cut. Now we're 19mm from the end of the 20mm arc, or 1mm total depth. Make sure to zero-out the X axis error of 1.805 before making the second cut!

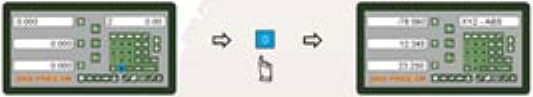


This is what the display should look like after the last cut has been made. Note the Z axis depth is 20mm, and the cutting tool is still "on the arc" because the X axis window shows an error of "0.000".

To temporarily return to the ABS coordinate system, press the "0" key.

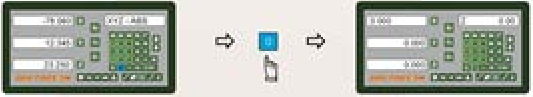
Presently in ARC function

Temporarily in ABS coordinate system



To return to the ARC function, press the "0" key.

Back in ARC function



Shrink Function

The display also has the capability to help with creating molds. When designing a mold, consideration for material shrinkage during cooling needs to be taken into account. For example, when molding in ABS material, the shrink factor is 1.009.

Normally, the mold designer has to manually calculate all of the reduced or expanded dimensions prior to the actual machining. The pitfalls of this method are the following:

1. It is a very time consuming process.

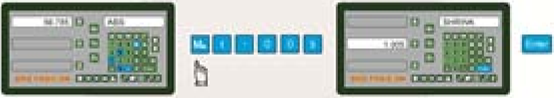
2. Because there a lot of calculations, it is almost inevitable that some mistake or omission may occur. Part of this is because there is no easy way to verify the final calculations.

3. Bear in mind the ever present need to keep project and development costs at a minimum. Simply put, the time and labor to verify a projects calculations may not be available.

The display provides a practical solution for mold makers with the "Shrinkage Calculation" function. This feature allows mold makers to easily calculate shrinkage and verify the expanded or reduced dimensions.

All shrinkage dimensions are actually multiples or divisions of the shrinkage factor, which is dependent on the planned mold material. Before machining, the operator must enter the shrink factor.

In our example, we're planning on using ABS plastic as our mold material, which as previously mentioned, has a shrink factor of 1.009.



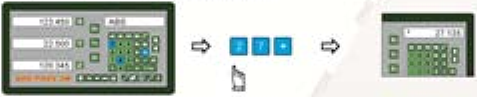
Shrinkage Calculations

The display provides a very easy to use shrink function, and allows the operator to easily calculate expanded or contracted dimensions.

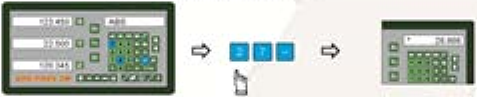
In the case where an operator needed to quickly verify an expanded or contracted measurement, shrink function has a quick and easy shrinkage calculator function.

Shrink function uses + for expanded computations and − for shrinkage computations.

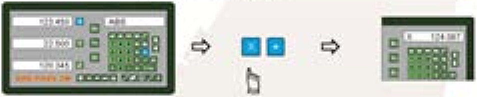
EXAMPLE: To calculate the expanded dimension of 27mm



EXAMPLE: To calculate the contracted or shrink dimension of 27mm



EXAMPLE: To calculate the expanded current axis dimension



Shrink Compensation

When using shrink compensation, the display automatically adjusts all measurements. There is no need to rewrite or recalculate workpiece dimensions.

If the operator insists on calculating individual shrink dimensions, the shrinkage compensation function can still be used as a very efficient way to manually verify the calculated dimensions.

Shrink function uses + for expanded compensations and − for shrinkage compensations.

"Expanded" Shrink

To view the expanded shrinkage compensations, such that the display is showing the expanded or calculated dimensions, use the following procedure:

Actual Dimensions

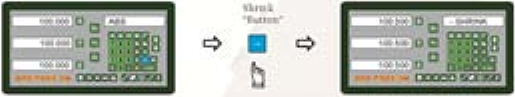
Compensated Dimensions
(Actual Dimensions x 1.005)

In order to remind the operator that the axis position has been "expanded" by the shrink factor, the display will differentiate the display in the following ways:

The display flashes "+ shrink"
The display will beep every 10 seconds
All functions and function keys are disabled

"Contracted" Shrink

To view the contracted shrinkage computations, such that the display is showing the contracted or calculated dimensions, use the following procedure:



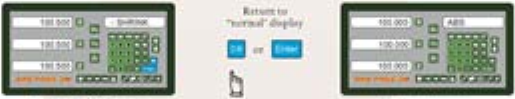
Actual Dimensions

Compensated Dimensions
(Actual Dimensions / 1.005)

In order to remind the operator that the axis position has been "contracted" by the shrink factor, the display will differentiate the display in the following ways:

The display flashes "- shrink"
The display will beep every 10 seconds
All functions and function keys are disabled

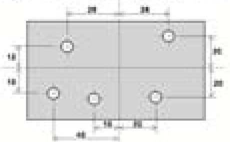
When in shrink compensation mode, if the operator desires to return to the "normal" or "uncorrected" display:



Compensated Dimensions
(Actual Dimensions / 1.005)

Actual Dimensions

EXAMPLE: Drilling holes in a plastic injection mold



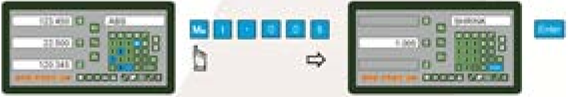
Because plastic material shrinks after it cools down after the plastic injection process, the dimensions of the holes in the mold have to be expanded just as the dimensions of the object.

Because the shrink factor is automatically applied to the entire object, the operator can simply drill at the dimensions as specified in the drawing. There is no need to separately compute distances for drill points.

Shrink Function - Example

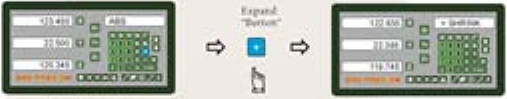
EXAMPLE: Entering a shrink factor

Our example project requires us to create a mold for ABS plastic, which has a shrink factor of 1.005.



First, let's set the display to read expanded compensation. To do this, push the "plus" key. In order to remind the operator that the axis position has been "expanded" by the shrink factor, the 3M5 will differentiate the display in the following ways:

The display flashes "+ shrink"
The display will beep every 10 seconds
All functions and function keys are disabled

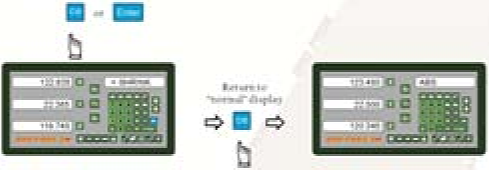


Actual Dimensions

Compensated Dimensions
(Actual Dimensions x 1.005)

At this point the operator can simply drill the holes as needed, no additional calculation is needed.

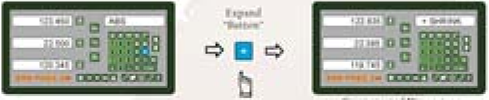
When the display is in shrink compensation mode, if the operator desires to return to the "normal" or "uncorrected" display



Compensated Dimensions
(Actual Dimensions x 1.005)

Actual Dimensions

After verifying the dimensions and a return to shrink compensated mode is desired



Actual Dimensions

Compensated Dimensions
(Actual Dimensions x 1.005)

At this point the operator can simply drill the holes as needed, no additional calculation is needed.

Parameters Setup

System Reset

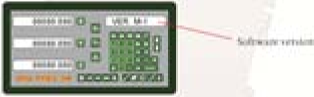
Every display is configured at the factory to run "out of the box". Normally, a total system reset will never be necessary. However, there may be a few instances where a reset of the system logic is desired. For example, if a number of users have set different system parameters and it's unknown what has been changed, or possibly a new user simply wishes to reset all settings back to the factory default. In any case, the method to change the display back to the default settings is as follows:

System Reset Procedure

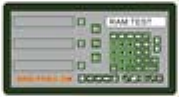
Switch off the display.

Switch on the display. As it powers up, the upper right message window displays the software version number, typically "VER. M-1". As this message is displayed, press the number "8" key to reset the display to the factory default settings.

Immediately after power-up, the display performs a self test ⇨

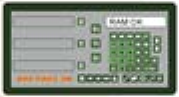


As the software version is displayed, press the 8 key to reset the display to the factory default settings.



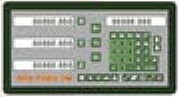
"RAM TEST" indicates memory test in progress

⇨



"RAM OK" indicates memory test in progress

"RESET" indicates memory test is complete



After the test is complete, the display proceeds to run an endless LED test to check for any missing LED segments. As soon as it is determined there are no missing segments, go ahead and shut off the display.

Parameters Setup

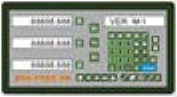
As mentioned before, every display is configured at the factory to run "out of the box". Entering and changing system parameters will not normally need to be accomplished. However, if a need arises, it is fairly simple and need only be accomplished once. For example, a common reason to enter the parameters setup menu would be to change the direction a scale is reading. Once set, the change in scale read direction is retained in permanent memory and need not be entered again, regardless of whether the machine is turned off again or not.

The Parameters Menu contains the following options:

AXIS NO	Enables the user to choose how many scale inputs are active.
DIRECTN	Permits the operator to change which direction the scale reads.
RESOLU	User selectable scale resolution.
RAD / DIA	Enables scales to read in either radius (RAD) or diameter (DIA) mode.
LIN COMP	Allows the operator to change / modify linear error compensation.
NL ERROR	Non-linear error compensation for pinpoint scale accuracy.
Z DIAL	Only for 2 axis readouts, and only used for ARC machining with a simulated third axis. Specifies the movement of the milling machine for each revolution of the Z axis handwheel.
DIAL INC	Only for 2 axis readouts, and only used for ARC machining with a simulated third axis. Specifies the Z axis handwheel increment.
R MODE	Only for 2 axis readouts, and only used for ARC machining with a simulated third axis. Allows the operator to choose between "MAX CUT" and "Z STEP". "MAX CUT" calculates a smoother ARC and results in a smoother finish. "Z STEP" is quicker, but leaves a rougher finish.
FLTR. PR	Filter function. Prevents distracting toggling of display, is especially during grinding.
QUIT	Exits the Parameters Setup menu and saves any changes.

How to Enter Parameters Setup

Switch on the display. As it powers up, the upper right message window displays the version number, typically "VER M-1". As this message is displayed, press the "enter" key.

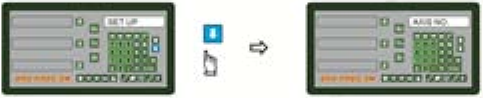


"SET UP" indicates the display has entered the Parameters Setup menu.

To scroll through the Parameters Setup menu, simply push the up or down arrow button(s).

QUIT • AXIS NO. • DIRECTN • RESOLU • RAD/DIA • LIN COMP •

NL ERROR • Z DIAL • DIAL INC • R MODE • FLTR. PR • QUIT •



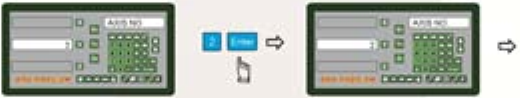
"AXIS NO." allows the operator to choose which axis scales will be used.

Selecting Axis Number

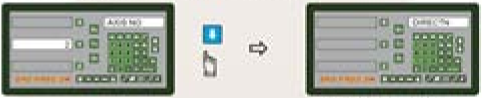
Press Enter to enter into the "Axis No." parameter setup. Note the default "3" displayed in the Y axis window.

To change how many scales will be utilized, press 1, 2 or 3 followed by the enter button. Note the default "3" displayed in the window will change to a "1", "2" or "3" depending on your selection.

To change the display from a 3 axis display to a 2 axis display, press "2", followed by the enter key.



To exit the Display Axis function, press the down arrow key.

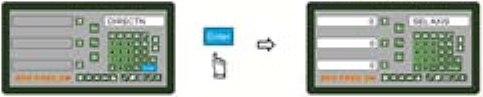


"DIRECTN" indicates the display is temporarily paused at the Direction parameter menu.

Changing Scale Read Direction

Press [Enter] to enter into the "Direction" parameter setup

Note the default "0" displayed in each X, Y, and Z axis window



To change the direction a scale reads, press the corresponding X, Y or Z axis button. Note the default "0" displayed in the window will change to a "1". Pressing the button again will change the display back to the default "0" and return the scale to the original direction of reading.

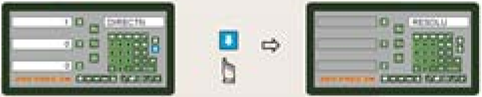
Press [X] or [X₀] to change the X axis scale read direction.

Note the display changes from a "0" to a "1". This indicates the scale read direction has been changed.

Note the "1" displayed in the X axis window

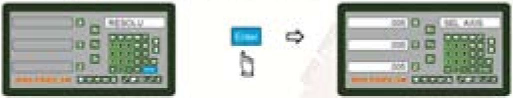
Now press [Enter] to exit from the direction function and return to the main menu.

Press [↓] to proceed to the next item in parameters setup, RESOLU, or Scale Resolution.

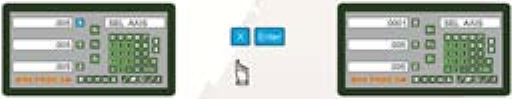


Scale Resolution

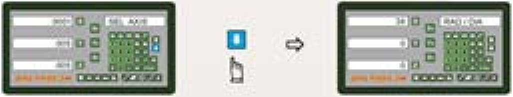
Press Enter to select the "RESOLU" or Scale Resolution function.



Scale resolution is selectable between either .005mm (.0002") or .001mm (.00005"). Selecting the X, Y or Z keys toggles between these two values. In this example, we are setting the X axis to .005mm resolution.

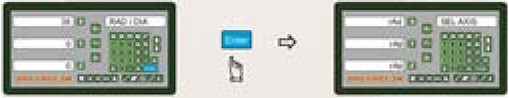


Press the down arrow key to exit from the Linear Error Compensation function and proceed to the next function, RAD / DIA or "Radius / Diameter".



Radius / Diameter (RAD / DIA) Function

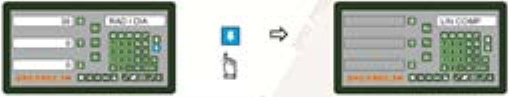
Press Enter to select the RAD / DIA function.



Pressing X or X₀ changes the X axis scale from radius to diameter mode and back again.

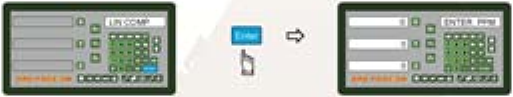
Note the display changes from "rAd" to "dIA". This indicates whether the scale reads in either radius or diameter mode. Normally, radius is the default value. When finished choosing the scale mode, press the enter key to accept the selection.

Press ↓ to continue from the Radius / Diameter function to the next function, "LIN COMP" or Linear Error Compensation.



Linear Error Compensation

Press Enter to select the "LIN COMP" or Linear Error Compensation function.



Linear compensation is specified in Parts Per Million (PPM). The method of calculation is as follows:

1. Measure the error using a step gauge or similar device, ie gauge block, of an accuracy level one grade higher than the measuring step. In other words, if you're measuring a 5 um scale, the measuring device resolution should be 1 um or better.

2. Record the error in microns (um). In this example, we recorded an error of 19um over a length of 500mm.

3. Project the error over a 1 meter length (1,000mm). For this example, the projected error would be:

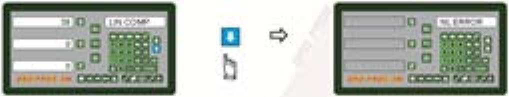
   19um x (1,000/500) = 38 um

4. Note the direction of error. For example, if the DRO display measured more or longer than the step gauge, the compensation value needs to be negative in order to reduce the displays value. In our example, the display indicated less than the step gauge, therefore our linear correction needs to be positive.

5. To enter a compensation value of 38 for our X axis, proceed as follows:

6. Congratulations, the Linear Error Compensation value of 18 has been entered!

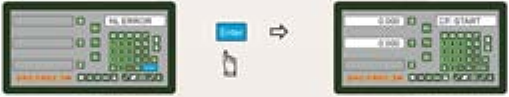
Press [↓] to continue from Linear Error Compensation to the next function, "NL ERROR" or Non-Linear Error Compensation.



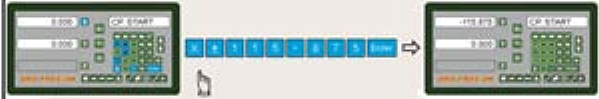
Non-Linear Error Compensation

Non-Linear error compensation is intended to maximize scale accuracy. All mechanical measuring systems are inherently flawed, however minutely. As a consequence of being physically manufactured, slight non-linear variations occur along the length of the scale. Non-linear error compensation attempts to compensate for these flaws by adding or subtracting a correction factor paired to the individual scale. By first measuring the scale against a known, extremely precise standard, an error "curve" is generated which defines the magnitude of error along the scale as a function of relative position. Once these values are input to the display, a non-linear error correction factor is automatically applied as the scale is moved, thereby greatly increasing scale accuracy.

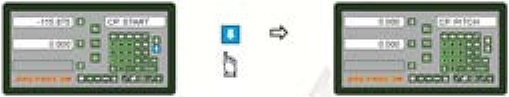
Press [Enter] to enter the Non-Linear Error function.



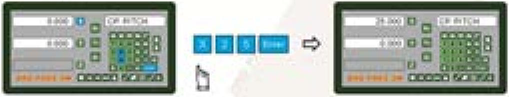
"CP START" represents the start position of where you began measuring the error, relative to the scales end or mid point. In this case, the error calculations were started 115.875mm to the left of the scales mid point, thereby making CP START = -115.875



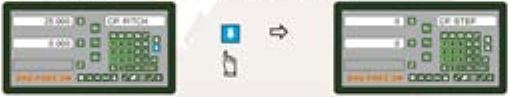
Next, press the down arrow key to move to "CP PITCH", which is the compensation profile pitch, or the transducer leg space



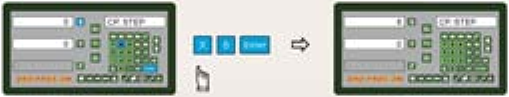
In this case, the leg space is a standard 25mm. Enter that value.



Next, press the down arrow key to move to "CP STEP", or compensation profile step.

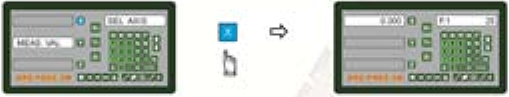


Compensation profile step is found by dividing the total distance traveled while measuring the error, divided by the pitch. In this case, it would be 200/25 = 8. Enter that value.

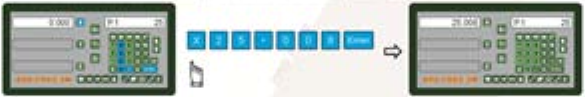


Next, press the down arrow key to move to "MEAS. VAL", where you will be prompted to enter the precise measured values.

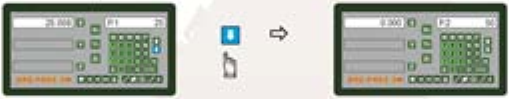
Press [X] to select the X axis.



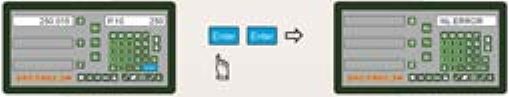
Now enter the measured values for each point. Our first measurement for point 1 was 25.008.



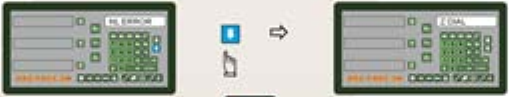
Arrow down for the next point.



Continue entering points until you reach your last correction point. In this example, it was P.10 - when finished inputting the value, press the "Enter" key two more times to return to the top menu.

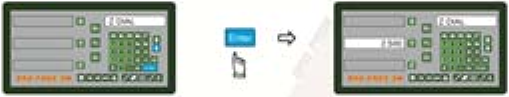


Press [↓] to continue to "Z Dial", the next function in the Set Up menu.

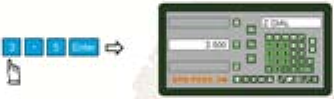


Z DIAL Function

Press Enter to select the "Z DIAL" entry function.



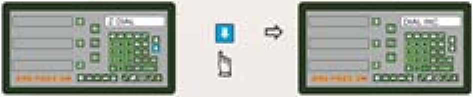
Z DIAL is only for 2 axis readouts, and is only used for ARC machining with a simulated third axis. Z DIAL specifies the movement of the milling machine for each revolution of the Z axis handwheel, which in our case we would like to change from the default value of 2.5mm to our custom value of 3.5mm.



The Z DIAL value of 3.5mm per handwheel revolution has been entered!

Press ⬇ to proceed to the next item in parameters setup, DIAL INC, or Dial Increment.

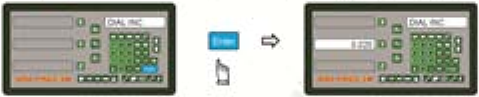
Dial Increment

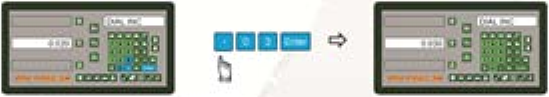


Dial Increment is only used with 2 axis readouts. DIAL INC stands for Dial Increment, and represents the dial increment calibration or 'step' of your machines Z axis handwheel. In our case we would like to change from the default value of .020mm to our custom value of .010mm.

Press Enter to select the "DIAL INC" entry function.

Here the Dial Increment has the default value setting of .020mm per increment. We would like to set .010 as the new increment.

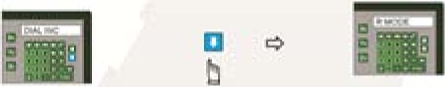




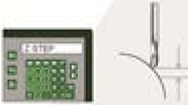
The Z DIAL increment of .01mm has been entered!

R Mode

Press ↓ to proceed to the next item in parameters setup, R MODE.



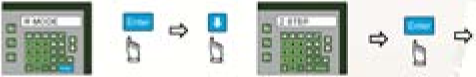
R MODE is only used with 2 axis readouts. It's purpose is to simulate a third axis when performing ARC machining into the Z axis. It allows the operator to choose between "MAX CUT" and "Z STEP". "MAX CUT" calculates a smoother ARC and results in a smoother finish. "Z STEP" is quicker, but leaves a rougher finish.



In Z STEP mode, the step is a fixed Z axis increment. While the process is quicker, the cut is not as smooth.

In MAX CUT mode, the "step" rotates around the cut, meaning it is more time consuming, but ultimately results in a much smoother finish.

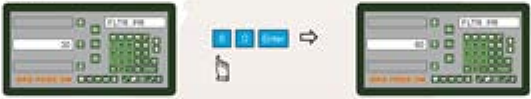
Press Enter to enter the R mode menu, then press ↓ or ↑ to select either the "2 STEP" or "MAX CUT" interpolation method. Press the "ent" key to confirm your selection. In this next example, we chose "2 STEP".



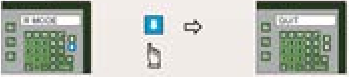
Press ↓ to proceed to the next item in parameters setup, "FLTR. PR".

Filter Process

"FLTR. PR" or Filter Process allows the operator to reduce distracting display "toggling" due to the inherent vibration present in certain operations, such as grinding. Inputting a larger value further dampens or reduces the inherent flicker or toggling of the last digits on the display, which increases accuracy and reduces operator fatigue.



Press ↓ to proceed to the next item in parameters setup, "QUIT".



Press the Enter button to exit from the Parameters Setup menu and save your changes.

Please note that in order to save any changes you have made in the parameters menu, you must exit the parameters menu via the "QUIT" command. Shutting off the display before using the QUIT command loses any changes you just entered.