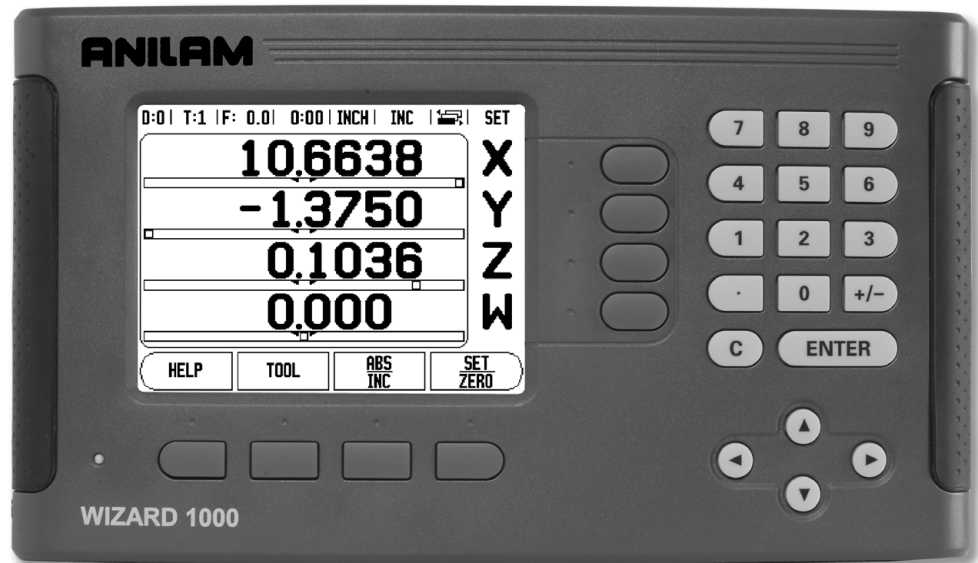


W1000 READOUTS

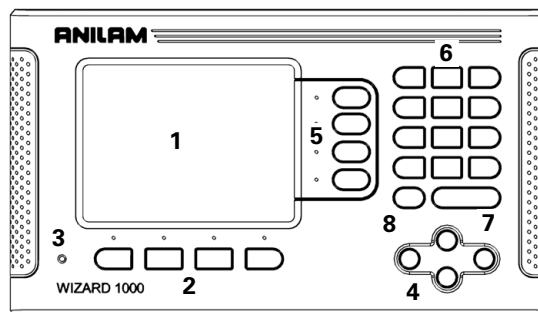


ANILAM

REFERENCE MANUAL

W1000 Key Layout

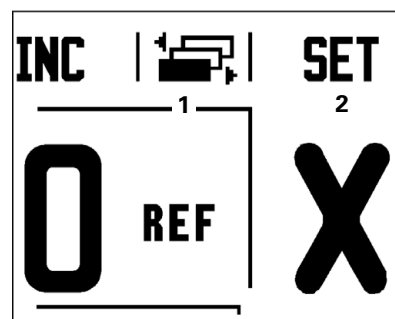
- 1 Display Area
- 2 Soft keys
- 3 Power Indicator light
- 4 Arrow Keys: Use the UP/DOWN keys to adjust the screen contrast.
- 5 Axis Keys
- 6 Numeric Keypad
- 7 ENTER key
- 8 CLEAR key



W1000 Soft keys



There are multiple pages of soft key functions to select from the operating modes. Use the LEFT/RIGHT arrow keys -4- to cursor through each page. The page indicator in the Status Bar will show the page orientation. The darkened page indicates the page you are currently on.





- 1 Page Indicator
- 2 Set Zero Indicator




Soft Key function (Page 1)	Soft key
Opens on-screen help instructions.	
Opens the Tool Table. Page 12 for Milling, and page 34 for Turning.	
Switches display between operating modes Actual Value (Absolute) / Distance-to-Go (incremental). See page 2.	
Toggles between Set/Zero functions. Used with individual axis keys (page 11).	

Soft Key function (Page 2a)	Soft key
Opens the Datum form to set the datum for each axis (page 14).	
Opens the Preset form. This form is used to set a nominal position. This is a Distance-To-Go (Incremental) function (page 17).	
Used to divide the current position by two (page 20).	

Soft Key function (Page 2a)	Soft key
Press to select the Circle Pattern, Linear Pattern, Incline Mill, or Arc Mill table (page 20).	
This soft key toggles between radius and diameter displays. This function is for Turning applications only (page 39).	

Soft Key function (Page 2b)	Soft key
Opens the Job Setup menu and provides access to the Installation Setup soft key (page 4).	
Press when ready to identify a reference mark (page 3).	
Opens the Calculator functions for standard math, trigonometry, RPM, and Taper for Turning functions.	
Toggles between inch and millimeter units (page 4).	

Soft Key function (Page 3)	Soft key
Selects the Program Mode (page 42).	

Readout Parameter Access Code

An access code must be entered before machine-related installation parameters can be set or changed. This prevents inadvertent adjustments to the installation setup parameters.



IMPORTANT!

The access code is 8891.

Access to Machine Parameter Operations

Refer to the Setup section also.



SETUP

Begin by pressing the SETUP soft key.

Press the soft key INSTALLATION SETUP.

Press the access code numbers **8891** using the numeric key pad.



ENTER

Press the ENTER key

The readout is now ready for machine parameter setting operations.



IMPORTANT!

To prevent setup parameters from being changed, remove this page from the Reference Manual after initially setting up the readout system. Retain this information in a safe place for future use.

Introduction

Software Version

The software version is shown on the initial power up screen.



This User's Manual covers the functions of the W1000 for both **milling**, and **turning** applications. Operational information is arranged in three sections: General Operations, Mill Specific Operations, and Turn Specific Operations.

W1000

DRO axis availability.



The W1000 Color DRO is available in **two, three, and four axis** form. The 4 axis W1000 DRO is used through out this manual for illustration, and description of function keys.

Symbols within Notes

Every note is marked with a symbol on the left indicating to the operator the type, and/or potential severity of the note.



General Information

e.g. on the behavior of the W1000.



Warning

e.g. when a special tool is required for a function.



Caution - Risk of electric shock

e.g. when opening a housing.

W1000 Fonts

The following shows how soft keys, and hard keys are represented within the text of this manual:

- Soft keys - SETUP soft key
- Hard keys - ENTER hard key



Hassle-Free Warranty

ACU-RITE Companies, Inc. (ACI) is proud to offer the 3-Year Hassle-Free Warranty for all digital readout systems, vision readout systems, and precision glass scales. This warranty will cover all of the ACI repair and replacement costs for any readout or precision glass scale returned during the three (3) year warranty period. ACI will repair or replace the damaged components - regardless of the product's condition absolutely free, no questions asked. This warranty covers both materials, and factory labor. In addition, authorized ACI service representatives will provide service labor (field service) for a period of one (1) year at no charge. Notice of the claimed defect must be received by ACI within the warranty period.

This warranty applies only to products and accessories installed and operated in accordance with this reference manual. ACI shall have no obligation, with respect to any defect, or other condition caused in whole or part by the customer's incorrect use, improper maintenance modification of the equipment, or by the repair, or maintenance of the product by any person except those deemed qualified by ACI.

Responsibility for loss of operation, or diminished performance due to conditions beyond ACI's control cannot be accepted by ACI.

The foregoing warranty obligations are in lieu of all expressed or implied warranties. ACU-RITE Companies, Inc. shall not be liable under any circumstances for consequential damages.

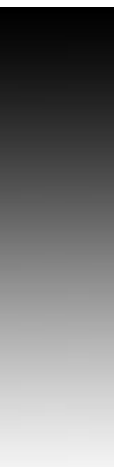


Table of Contents

W1000 Key Layout	i
W1000 Soft keys	i
Readout Parameter Access Code	
Access to Machine Parameter Operations	iii
Introduction	
Software Version	v
W1000	v
Symbols within Notes.....	v
W1000 Fonts	v
Hassle-Free Warranty	
I - 1 Introduction to the W1000	
Screen Layout	1
Operating Modes	2
Reference Mark Evaluation.....	3
Working without reference mark evaluation.....	3
Enable/Disable Ref function.....	3
Job Setup Parameters	4
Units.....	4
Scale Factor	4
Mirror	5
Edge Finder (milling applications only)	5
Diameter Axes	5
Measured Value Output.....	6
Near Zero Warning.....	6
Status Bar Settings	6
Job Clock	6
Skew Compensation (Milling applications only).....	7
Remote Switch	8
DRO View Settings	8
Console Adjustment	10
Language	10
Import/Export.....	10
Set/Zero Soft Key Details.....	11

I - 2 Milling Specific Operations

Soft Key Functions Detailed	12
Tool Soft Key	12
Tool Table	12
Tool Compensation	13
Sign for the length difference ΔL	13
Calling the Tool from the Tool Table	14
Datum Soft Key	14
Probing Functions for Datum Setting	14
Probing with a Tool	15
Example: Probe workpiece edge and set edge as datum	16
Presets.....	17
Absolute Distance Preset	17
Incremental Distance Preset	19
1/2 Soft Key	20
Circle, and Linear Patterns	20
Soft key Functions	21
Circle, and Linear Pattern Table Entry.....	22
Circle, and Linear Pattern Soft keys.....	23
Circle, or Linear Pattern Execution	24
Example: Enter data and execute a circle pattern.	25
Incline, and Arc Milling.....	27
Soft Key Functions.....	27
Incline, and Arc Milling Table Entry.....	28
Incline, and Arc Milling Table Entry.....	29
Arc Milling.....	30
Incline and Arc Mill Execution.....	31
Z/W Coupling (4 axes Milling)	32
Enabling Z/W Coupling.....	33
Disabling Z/W Coupling.....	33

I - 3 Turning Specific Operations

Tool Display Icon.....	34
Tool Table	34
Datum Setting.....	36
Taper Calculator Soft Key	38
Presets.....	39
Radius/Diameter Soft Key.....	39
Vectoring.....	40
Z Coupling.....	40
Disabling Z0, and Z Coupling	41

I - 4 Programming W1000	
Program Mode Soft key functions	43
View Soft Key	44
Feature Soft Key functions	45
Program Function Soft keys.....	48
Step Function Soft keys.....	48
Editing, and moving through a program.....	49
I - 5 Executing a Program	
Run Soft Keys	50
Executing Steps.....	50
II - 1 Installation Setup	
Installation Setup Parameters.....	53
Encoder Setup	53
Display Configuration.....	55
Error Compensation.....	55
Linear Error Compensation	56
Automatic Linear error compensation.....	56
Non-Linear Error Compensation.....	57
Starting a Non-Linear Error Compensation Table	57
Configuring the Compensation Table.....	58
Automatic Non Linear error compensation	58
Backlash Compensation	59
Serial Port	60
Counter Settings.....	60
Diagnostics	61
Keypad Test.....	61
Edge Finder Test.....	61
Display Test	61
II - 2 RS-232C Serial Communications	
Serial port.....	62
II - 3 Installation and Electrical Connections	
Installation.....	63
Electrical requirements	63
Environmental.....	63
Preventative maintenance	63

II - 4 I/O Connections
 Wiring the Serial communication cable 65

II - 5 Remote Switch Data Output
 Data output using external signals..... 66
 Data output using Edge Finder 68

II - 6 Dimensions
 DRO Dimensions 71

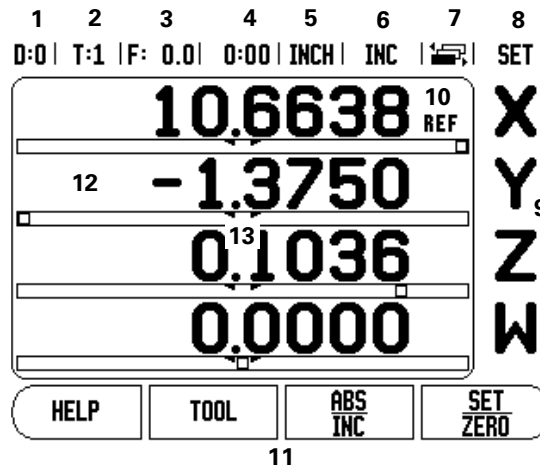
I - 1 Introduction to the W1000

Screen Layout

- 1 Datum
- 2 Tool
- 3 Feed Rate
- 4 Job Clock
- 5 Unit of Measure
- 6 Operating Modes
- 7 Page Indicator
- 8 Set/Zero
- 9 Axis Labels
- 10 Ref Symbol
- 11 Soft key Labels
- 12 Display Area
- 13 Near Zero Warning (In Distance-To-Go mode only)

Anilam's W1000 color readout provides application-specific features that allows you to obtain the most productivity from your manual machine tools.

- **Status Bar** - This displays the current datum, tool, feed rate, job clock time, unit of measure, operating mode status, page indicator, and set/zero. See Job Setup for details on setting up the Status Bar parameters.
- **Display Area** - Indicates the current position of each axis. Also shows forms, fields, instruction boxes, error messages and help topics.
- **Axis Labels** - Indicates axis for corresponding axis key.
- **Ref Symbols** - Indicates current reference mark status.
- **Soft key Labels** - Indicates the various milling or turning functions.



Operating Modes

The W1000 has two operating modes: **Distance-To-Go** (INCREMENTAL), and **Actual Value** (ABSOLUTE). The Actual Value mode always displays the current actual position of the tool, relative to the active datum. In this mode, all moves are done by traveling until the display matches the nominal position that is required. The Distance-To-Go feature enables you to approach nominal positions by traversing to display value zero. When working within the Distance-To-Go mode you can enter nominal coordinates as either absolute or incremental dimensions.

Milling applications have only the tool length offsets active while in the Actual Value Mode. In the Distance-To-Go mode, both the diameter and length offsets are used to calculate the amount of "distance-to-go" required to get to the desired nominal position. This is relative to the edge of the tool that will be doing the cutting.

Turning applications have both diameter and length offsets in both the Actual Value and Distance-To-Go modes.

Press the ABS/INC soft key to toggle between these two modes. To view other soft key functions in either Actual Value or Distance-To-Go mode, use the LEFT/RIGHT arrow keys.

The Milling application provides a quick method for coupling the Z and W axes position on a 4 axis system. For more information, see "Z/W Coupling (4 axes Milling)" on page 32.

The Turning application provides a quick method for coupling the Z axes positions on a 3 axis system. For more information, see "Z Coupling" on page 40.



Reference Mark Evaluation

The W1000 reference mark evaluation feature (1), automatically re-establishes the relationship between axis slide positions and display values that was last defined by setting the datum.

For each axis with an encoder that has reference marks, the “REF” indicator will flash for that axis. After crossing over the reference marks, the indicator will stop flashing and change to a non-flashing “REF” indicator.

Working without reference mark evaluation

The W1000 can also be used without crossing over the reference marks.

- ▶ Press the NO REF soft key to exit the reference mark evaluation routine, and continue.

The W1000 can still cross over reference marks at a later time. Such as if it becomes necessary to define a datum that can be re-established after a power interruption.

- ▶ Press the ENABLE REF soft key to activate the position recovery routine. Press the RIGHT / LEFT arrow key if the soft key is not shown on current screen.

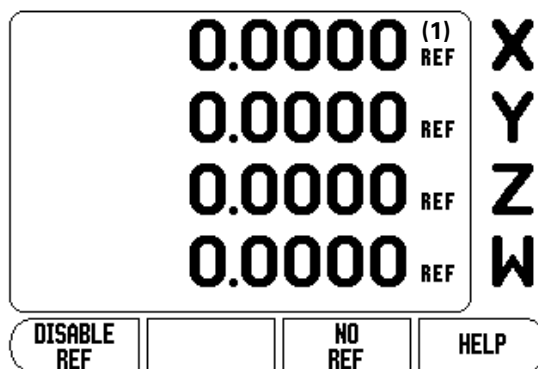


If an encoder is setup without reference marks, then the REF indicator will not be displayed, and a datum set from any axis will be lost once power is turned off.

Enable/Disable Ref function

The ENABLE/DISABLE soft key, that is present during the position recovery routine, allows the operator to select a specific reference mark on an encoder. This is important when using encoders with Fixed Reference Marks (instead of ones with Position-Trac™ feature). When the DISABLE REF soft key is pressed, the recovery routine is paused and any reference marks that are crossed during encoder movement are ignored. When the ENABLE REF soft key is then pressed, the recovery routine once again becomes active and the next crossed reference mark will be selected.

D:0 | T:1 | F: 0.0 | 0:00 | INCH | ABS | |



Not all reference marks have to be crossed over on each encoder, only those that are needed.

- ▶ Once reference marks for all desired axes are established, press the NO REF soft key to cancel out of the routine. If all reference marks have been found, the W1000 will return to the DRO display screen automatically.

Job Setup Parameters

- ▶ To view and change Job Setup parameters, first press the SETUP soft key. This is the screen that will now be on the DRO.
- ▶ Use the UP/DOWN arrow keys to highlight the parameters of interest.
- ▶ press the ENTER key.

Units

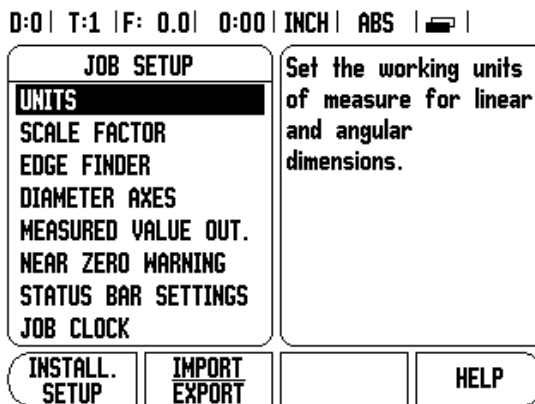
The Units form is used to specify the preferred display units and format. You can also select the unit of measure by pressing the INCH/MM soft key in either Actual Value or Distance-To-Go mode.

Scale Factor

The scale factor may be used to scale the part up or down. A scale factor of 1.0 creates a part with the exact size as dimensioned on the print. A scale factor >1 "grows" the part, and <1 "shrinks" the part.

The following probing soft key functions are available:

- The numeric keys are used to enter a number greater than zero. The number range is 0.1000 to 10.000. A negative value may also be entered.
- The scale factor settings will be retained on a power cycle.
- When the scale factor is a value other than 1, the scaling symbol ∇ is shown on the axis display.
- The ON/OFF soft key is used to disable the current scale factors.



Mirror



A scale factor of -1.00 will produce a mirror image of the part. You can both mirror and scale a part at the same time.

Edge Finder (milling applications only)

The diameter, length offset and units of the edge finder are set in this form. Both values are in the units indicated in the form. Please see "Probing Functions for Datum Setting" on page 14 for details on using Edge Finder functions.

- The numeric keys are used to enter values of diameter and length. The diameter must be greater than zero. The length is a sign value (negative, or positive).
- A soft key is provided to indicate the units of measure for the edge finder.
- The edge finder values will be retained on a power cycle.

Diameter Axes

Select Diameter Axes to bring up the diameter screen shown here to set which axes can be displayed in either radius or diameter values. ON indicates that the axis position will be displayed as a diameter value. When OFF, the Radius/Diameter feature does not apply. For turning applications see "Radius/Diameter Soft Key" on page 39 for the Radius/Diameter feature.

- ▶ Cursor to Diameter Axes, and press ENTER.
- ▶ The cursor will be in the X axis field. Depending on the parameter you need for that axis press ON/OFF soft key to turn feature on or off.
- ▶ Press ENTER.

D:0 | T:1 | F: 0.0 | 0:00 | INCH | ABS | |

DIAMETER AXES	
X	OFF
Y	OFF
Z	OFF
W	OFF

Set to ON to display the position as a diameter value.

ON OFF HELP

Measured Value Output

With the measured value output feature, probe measurement values can be sent over the serial port. Also output of the current display positions is activated via a command (Ctrl B), sent to the W1000 over the serial port.

The Measured Value Output form is used to set data output during probing operations.

- Data Output Probing (Milling Only), may be set to either On, or Off. When On, the measurement data is output when the probe operation is completed.

Refer to chapter "II - 5 Remote Switch Data Output on page 66" for information on the format of the output data.

Near Zero Warning

The Near Zero Warning form, is used to configure the bar graph. This is shown below the axes' display in Distance-To-Go mode. Each axis has its own range.

- ▶ Press the ON/OFF soft key to enable, or begin entering values using the numeric keys. The current position box will begin moving when the position is within range.

Status Bar Settings

The Status Bar is the segmented bar at the top of the screen which displays current datum, tool, feed rate, job clock and page indicator.

- ▶ Press the ON/OFF soft key for each setting you want to see displayed.

Job Clock

The job clock shows the hours (h), minutes (m), seconds (s). It operates like a stop watch showing elapsed time. The clock starts timing from 0:00:00.

- The elapsed time field shows the total accumulated time from each interval.
- ▶ Press the START/STOP soft key, the status field will read RUNNING. Press it again to stop time from elapsing.

- ▶ Press RESET to reset the elapsed time. Resetting will stop the clock if it is running.



Pressing the Decimal key while in operating mode, will also stop and start the clock. Pressing the Zero key will reset the clock.

Skew Compensation (Milling applications only)

Skew compensation allows the user to drill holes on a workpiece without aligning the workpiece on the machine. Use this feature only for drilling holes.

The Skew Compensation setup form is found under Job Setup. The form is used to turn skew compensation on, or off; to set, or learn the skew angle; and to set the datum.

- ▶ In the Status Field, press the **ON/OFF** soft key, turn skew compensation on, or off.
- ▶ In the Angle Field, enter the skew angle if known. To learn the angle, use the edge finder (or a tool) to touch off the part.



The skew angle is learned by touching off two points along one side. When using the edge finder, the location of the edge is captured automatically.

- ▶ When using a tool, touch the tool to the edge of the part, and press the TEACH soft key.

Setting the datum

- After learning the angle, the datum may be set to the corner of the part by touching off one point along the other side see "Datum Soft Key" on page 14. In the datum calculation, the system compensates for the radius of the edge finder (or current tool).

The readout automatically compensates for the misaligned workpiece on the machine when running a program, a hole pattern, or preset. Move the machine until the X axis and the Y axis indicate zero.



When skew compensation is activated, the skew icon is displayed on the right side of the axis display.

Remote Switch

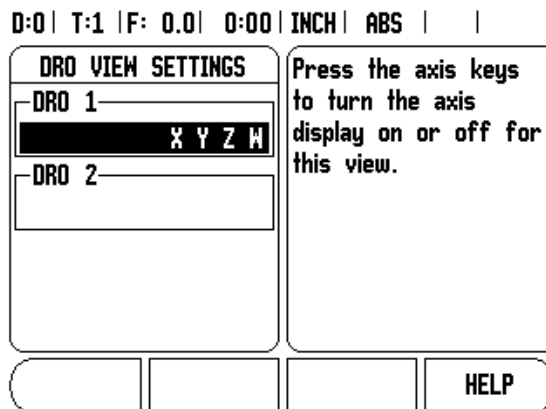
The **remote switch** sets the parameters so the external switch (pendant, or foot switch) can be enabled to perform any or all of the following functions: Data Output; Zero, and Next Hole. Refer to Section II for information on connecting remote switches via Grounding Edge Finder input.

- Data Output - to send position information out of the serial port or to print the current position.
- Zero - to zero one or more axes. In Distance-To-Go mode, it will zero the Distance-To-Go display. In Actual Value mode, it will zero the datum).
- Next Hole key press will move to the next hole within a pattern (i.e. Hole Pattern).
 - ▶ While in the Data Output Field, press the ON/OFF soft key to ON to send the current position over the serial port when the switch is closed.
 - ▶ While in the Zero Field, press the appropriate axis keys to enable or disable zeroing of the axis display positions when the switch is closed.
 - ▶ While in the Next Hole Field, press the ON/OFF soft key to ON to move to the next hole within a pattern.

DRO View Settings

The W1000 has the ability to define up to two DRO display configurations (views). Each view defines which axes displays appear on the screen when the view is selected.

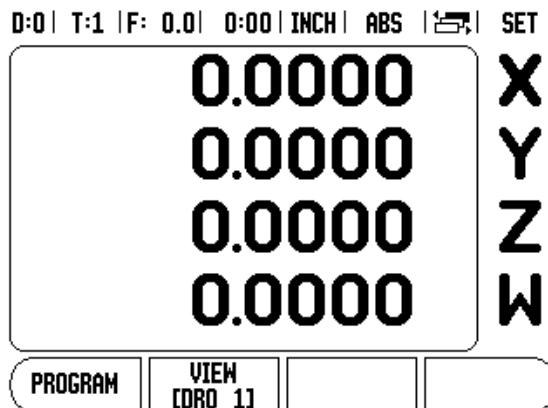
With two views available, one may be set to all available axes, and the other may be a subset of the axes.



When two DRO views are configured, the DRO mode includes a VIEW soft key which is used to toggle between DRO 1 and DRO 2 views. The current view (1 or 2) is indicated on the soft key.

To setup the DRO views, press SETUP, cursor to DRO view settings, and press ENTER. The configuration form will open.

Press the axis key to turn that axis display on. If an axis key is pressed on an axis that is already on, it will turn off. The default configuration is that all available axes are on for DRO 1, and all axes are off for DRO 2. The W1000 requires that at least one axis must be turned on at all times. Axis view settings are remembered when power is cycled.



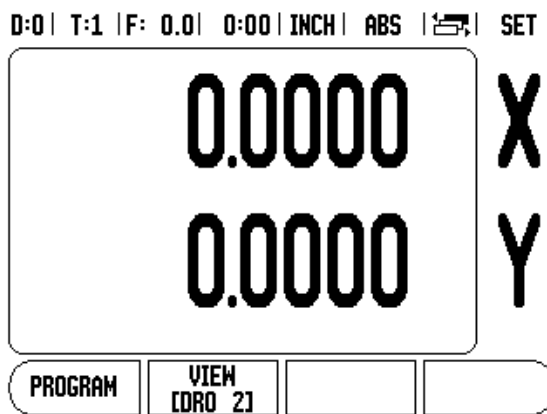
Multiple DRO views are only available when the full screen DRO is displayed. In screens where the small DRO is displayed, all axes positions are displayed and multiple DRO views are not available.

When running a program or a single cycle pattern, the large DRO uses the current DRO view that is selected. The view with the graphic and small DRO will show all available axes.



The size of the position values, and the axis labels are adjusted based on the number of axes being displayed. Smallest at 4 axes, largest at 1, or 2 axis display.

Axis keys correspond to the axes being displayed in the current DRO view. If only two axes are being displayed, and there are more axis keys (3 or 4), then only the top two keys will function. Unmatched keys are ignored.



Console Adjustment

The LCD's brightness and contrast can be adjusted either by using the soft keys in this form, or by using the UP/DOWN arrow keys on the keypad in either operating mode. The settings of brightness, and contrast may need to be adjusted due to variations in ambient lighting, and operator preference. This form is also used to set the display saver's idle time-out. The display saver setting is the amount of time the system is idle before the LCD is turned off. The idle time may be set from 30 to 120 minutes. The display saver can be disabled during the current power cycle.

Language

The W1000 supports multiple languages. To change the language selection:

- ▶ Press the LANGUAGE soft key until the desired language selection appears on the soft key, and the form.
- ▶ Press ENTER to confirm your selection.

Import/Export

Job, and Installation Setup parameters can be imported or exported over the "Regular Type B" USB port.

- ▶ Press the IMPORT/EXPORT soft key in the Setup screen.
- ▶ Press IMPORT to download operating parameters from a PC.
- ▶ Press EXPORT to upload the current operating parameters to a PC.
- ▶ Press the C key to exit.

Set/Zero Soft Key Details

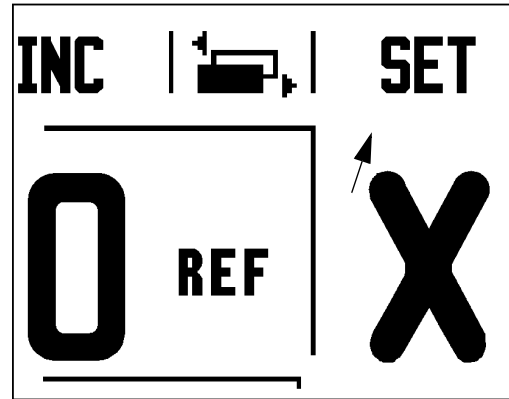
The SET/ZERO soft key is a key that determines the effect of pressing an Axis key. This key is a toggle key, switching the functionality between Set, and Zero. The current state is indicated in the Status Bar shown as "SET" in this view.

When the state is Set, and the W1000 is in Actual Value mode, selecting an Axis key opens the Datum form for the selected axis. If the W1000 is in Distance-To-Go mode, a Preset form opens.

When the state is Zero, and the W1000 is in Actual Value mode, selecting an Axis key sets the current datum for that axis to zero, at the current position. If it is in Distance-To-Go mode, the current Distance-To-Go value is set to zero.



If the W1000 is in Actual Value mode and the state of Set/Zero is zero, pressing any Axis key resets the current datum to zero at the current location for that axis.



I - 2 Milling Specific Operations

This section discusses operations and soft key functions specific to milling applications only.

Soft Key Functions Detailed

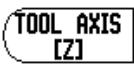




Tool Soft Key


This soft key opens the tool table and provides access to the Tool form for entering a tool's parameters (a soft key is used on a one axis readout). The W1000 can store up to 16 tools within the tool table.

Tool Table

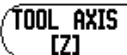



The W1000 tool table provides a convenient way to store diameter and length offset information for each of the tools that may be commonly used, up to 16 tools. An example of the Tool Table screen is shown here

The following soft keys are also available while in the Tool Table form or in the individual tool data form:

Function	Soft key
Use this key to select which axis all the tool length offsets will affect. The tool's diameter values will subsequently be used to offset the remaining two axes.	
Press to automatically enter the tool offset length. This key is only available in the TOOL LENGTH field.	
The Tool Types form is opened using this key, and is only available in the Type field.	
To remove a tool from the table, use the CLEAR TOOL key.	
To use a tool from the table, highlight the tool, and press the USE TOOL key.	

D:0 | T:1 | F: 0.0 | 0:00 | INCH | ABS |  |

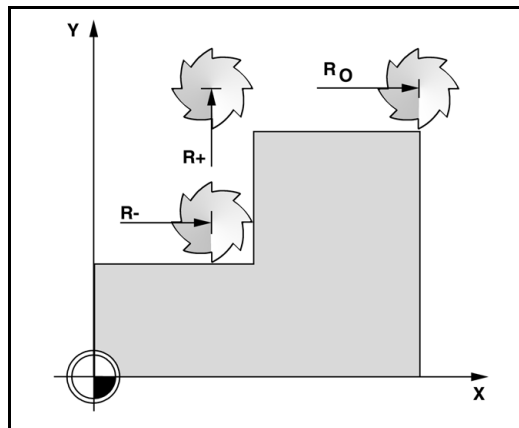
TOOL TABLE (DIA/LENGTH)			
1	2.000/	20.000 MM	DRILL
2	5.000/	14.000 MM	PILOT DRL
3	25.000/	5.000 MM	CTR-BORE
4	6.000/	12.000 MM	CARB ML
5	10.000/	25.000 MM	BROACH
6	2.000/	0.000 MM	FL END ML
7	2.500/	0.000 MM	FL END ML
8	3.000/	5.000 MM	

			
--	---	---	---

Tool Compensation

This allows you to enter workpiece dimensions directly from the drawing. **R** represents the tool radius, and examples of the **R** values are shown in this figure. The displayed distance to go is then automatically lengthened **R+**, or shortened **R-** by the value of the tool radius. For more information see "Presets" on page 17.

The length offset may be entered as a known value or the W1000 may determine the offset automatically. The tool length is the difference in length ΔL between the tool and the reference tool. The length difference is indicated with the " Δ " symbol. The reference tool is indicated by T1.



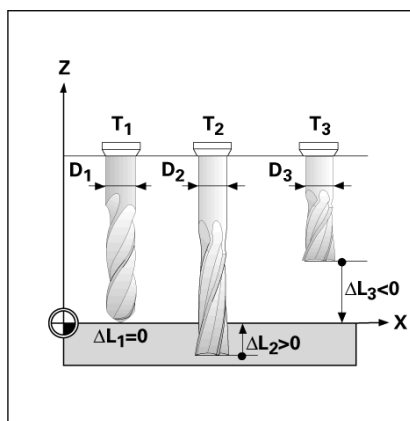
Sign for the length difference ΔL

If the tool is **longer** than the reference tool: $\Delta L > 0 (+)$.

If the tool is **shorter** than the reference tool: $\Delta L < 0 (-)$.

The length offset may be entered as a known value, or the W1000 may determine the offset automatically. To have the W1000 determine a tool's length offset involves touching the tip of each tool to a common reference surface. This allows the W1000 to determine the difference between the length of each tool.

Move the tool until it's tip is touching the reference surface. Press the TEACH LENGTH soft key. The W1000 will calculate an offset relative to this surface. Repeat the procedure for each additional tool using the same reference surface.



Only the tools set using the same reference surface may be changed without having to reset the datum.

Calling the Tool from the Tool Table

- ▶ To call a tool, press the TOOL hard key.
- ▶ Use the UP/DOWN arrow keys to cursor through the selection of tools (1-16). Highlight the tool you want.
- ▶ Verify the proper tool has been called, and press the TOOL, or C key to exit.

Datum Soft Key

Datum settings define the relationships between the axis positions, and the display values.

Setting datum points is done by using the W1000 probing functions either with a tool, or with an edge finder.

Probing Functions for Datum Setting

An electronic Edge Finder can be connected through the edge finder input. The W1000 also supports a grounding type edge finder connected via the 3.5mm Phono Jack on the back of the unit. Both types of edge finders operate the same way.

The following probing soft key functions are available:

- Workpiece edge as datum: EDGE soft key.
- Centerline between two workpiece edges: CENTER LINE soft key.
- Center of a hole or cylinder: CIRCLE CENTER soft key.

In all probing functions, W1000 takes into account the probes's entered tip diameter. During probing functions, with an electronic, or grounding edge finder, the display freezes with the location of the edge, centerline, or circle center.

- ▶ To abort the probing function while it is active, press the C key.

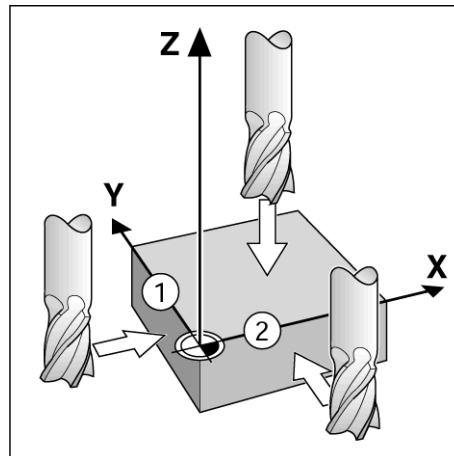


To perform probing, the probe's dimensional characteristics must first be entered into Job Setup. See "Job Setup Parameters" on page 4.

Probing with a Tool

If using a tool, or non-electrical edge finder to set datum points, the W1000 probing functions can still be used.

Datum points can be set by touching the edges of a workpiece, one after the other with a tool, and then manually entering the tool's position as datum points.

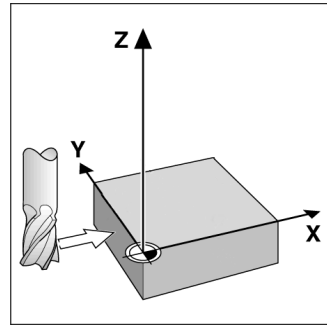


D:1 | T:1 | F: 0.0 | 0:00 | INCH | ABS | | SET

SET DATUM		X	0.0000	X
DATUM NUMBER		Y	0.0000	
1		Z	0.0000	
		W	0.0000	
DATUM		Enter the new actual position of the tool or press PROBE.		
X	-1.5000			
Y	-1.5000			
Z	0			
W				
PROBE				HELP

Example: Probe workpiece edge and set edge as datum

- ▶ Preparation: Set the active tool to the tool that will be used to set the datum. An end mill is shown in this example, along with the Set Datum screen.
 - Datum axis: X = 0
 - Tool diameter D = 0.25"
- ▶ Press DATUM.
- ▶ Press the Down arrow key until the X axis field is highlighted.
- ▶ Press PROBE soft key.
- ▶ Press EDGE soft key.
- ▶ Touch workpiece edge.
- ▶ Press the TEACH soft key to store the current absolute value while the tool is in contact with the workpiece edge. The location for the touched edge will take into account the diameter of the tool in use (T:1, 2...), and the last direction the tool was moved prior to pressing the TEACH soft key.
- ▶ Retract the tool from the workpiece and enter 0" and then press ENTER.



D:0 | T:1 | F: 0.0 | 0:00 | INCH | ABS | | SET

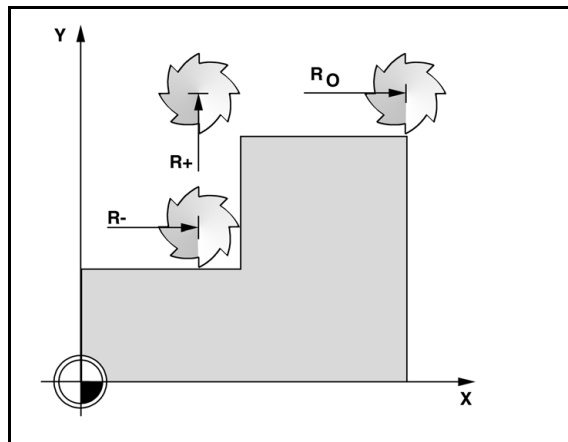
SET DATUM		X	0.0000	X Y Z W
DATUM NUMBER		Y	0.0000	
0		Z	0.0000	
DATUM		W	0.0000	
X	[REDACTED]	Select the probe function.		
Y				
Z				
W				

EDGE CENTER LINE CIRCLE CENTER HELP

Presets

The Preset function allows the operator to indicate the nominal (target) position for the next move. Once the new nominal position information is entered the display will switch to Distance-To-Go mode and show the distance between the current position and the nominal position. The operator now only needs to move the table until the display is zero and he will be at the required nominal position. The information for the location of the nominal position can be entered as an absolute move from the current datum zero or as an incremental move from the current nominal position.

Presetting also allows the operator to indicate which side of the tool will be doing the machining at the nominal position. The R+/- soft key in the Preset form defines the offset that will be in effect during the move. **R+** indicates that the center line of the current tool is in a more positive direction than the edge of the tool. **R-** indicates that the center line is in a more negative direction than the edge of the current tool. Using R+/- offsets automatically adjusts the distance-to-go value to account for the diameter of the tool.

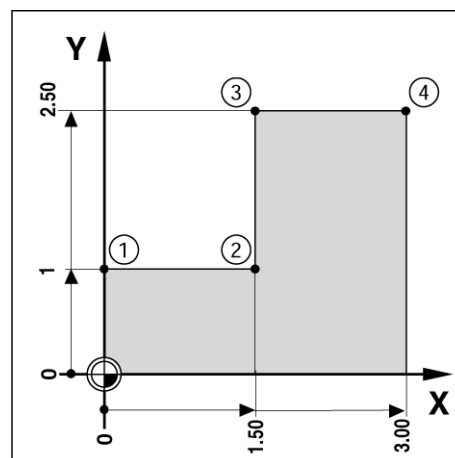


Absolute Distance Preset

Example: Milling a shoulder by traversing to display value zero using absolute position.

The coordinates are entered as absolute dimensions; the datum is the workpiece zero. Using this figure example:

- Corner 1: X = 0 / Y = 1
- Corner 2: X = 1.50 / Y = 1
- Corner 3: X = 1.50 / Y = 2.50
- Corner 4: X = 3.00 / Y = 2.50



Press the PRESET soft key, then an axis key to recall the last entered preset value for that axis.

Preparation:

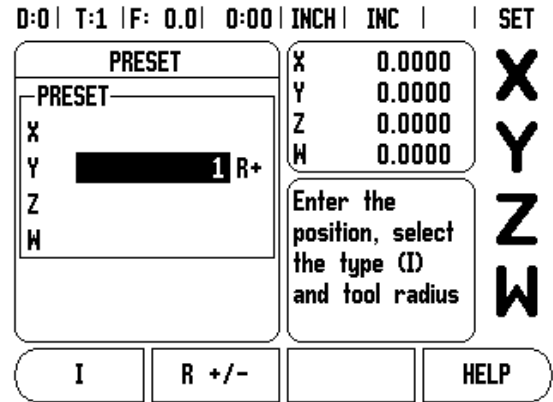
- ▶ Select the tool with the appropriate tool data.
- ▶ Pre-position the tool to an appropriate location (such as X = Y = -1").
- ▶ Move the tool to milling depth.
- ▶ Press the PRESET soft key.
- ▶ Press the Y axis key

-ALTERNATIVE METHOD -

- ▶ Press the SET/ZERO soft key so that you are in Set mode.
- ▶ Press the Y axis key.
- ▶ Enter nominal position value for corner point **1**: Y = 1", and select tool radius compensation R + with R+/- soft key. Press until R+ is shown next to axis form.
- ▶ Press ENTER.
- ▶ Traverse the Y axis until the display value is zero. The square in the near zero warning is now centered between the two triangular marks.
- ▶ Press the PRESET soft key.
- ▶ Press the X axis key.

-ALTERNATIVE METHOD -

- ▶ Press the SET/ZERO soft key so that you are in Set mode.
- ▶ Press the X axis key.
- ▶ Enter nominal position value for corner point **2**: X = 1.5" select tool radius compensation R - with R+/- soft key. Press twice until R- is shown next to axis form.
- ▶ Press ENTER.
- ▶ Traverse the X axis until the display value is zero. The square in the near zero warning is now centered between the two triangular marks.
- ▶ Presets can be entered in the same manner for corners 3, and 4.



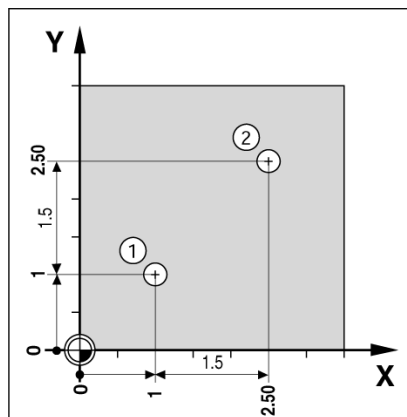
Incremental Distance Preset

Example: Drilling by traversing to display value zero with incremental positioning.



Enter the coordinates in incremental dimensions. These are indicated in the following (and on the screen) with a preceding **I** (Incremental). The datum is the workpiece zero.

- Hole **1** at: X = 1" / Y = 1"
- Distance from hole **1** to hole **2**: XI = 1.5" / YI = 1.5"
- Hole depth: Z = -0.5"
- Operating mode: DISTANCE-TO-GO (INC)
- ▶ Press the PRESET soft key.
- ▶ Press the X axis key.
- ▶ Enter nominal position value for hole **1**: X = 1" and ensure no tool radius is active. Note that these presets are Absolute Presets.
- ▶ Press the DOWN arrow key.
- ▶ Enter nominal position value for hole 1: Y = 1".
- ▶ Ensure no tool radius compensation is showing.
- ▶ Press the DOWN arrow key.
- ▶ Enter the nominal position value for the hole depth: Z = -0.5". Press ENTER hard key.
- ▶ Drill hole 1: Traverse the X, Y and Z axis until the display value is zero. The square in the near zero warning is now centered between the two triangular marks. Retract the drill.



D:0 | T:1 | F: 0.0 | 0:00 | INCH | INC | | SET

PRESET		X	0.0000	X Y Z W
PRESET		Y	0.0000	
XI	1.5000	Z	0.0000	
YI	1.5	W	0.0000	
Z		Enter the position, select the type (I) and tool radius		
W				
I		R +/-		HELP

To preset the location for Hole 2:

- ▶ Press the PRESET soft key.
- ▶ Press the X axis key.
- ▶ Enter nominal position value for hole **2**: $X = 1.5''$, mark your input as an incremental dimension, press the I soft key.
- ▶ Press the Y axis key.
- ▶ Enter nominal position value for hole **2**: $Y = 1.5''$, mark your input as an incremental dimension, press the I soft key.
- ▶ Press ENTER.
- ▶ Traverse the X and Y axes until the display value is zero. The square in the near zero warning is now centered between the two triangular marks.
- ▶ To preset the Z axis:
- ▶ Press the PRESET soft key.
- ▶ Press the Z axis key.
- ▶ Press the ENTER key (use last entered preset).
- ▶ Drill hole 2: Traverse Z axis until the display value is zero. The square in the near zero warning is now centered between the two triangular marks.
- ▶ Retract the drill.

1/2 Soft Key

The 1/2 hard key is used to find the centerline (or midpoint) between two locations along a selected axis of a workpiece. This can be performed in either Actual Value or Distance-To-Go mode.



This feature will change datum locations when in Actual Value mode.

Circle, and Linear Patterns








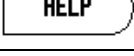
This section describes the Circle, and Linear pattern tables, and capabilities. The W1000 provides storage of 10 user definable patterns each for Circle, and Linear. Once patterns are defined, they are remembered when power is cycled. They can be recalled, and executed from the DRO, or from a program.

The View Graphic enables verification of the hole pattern before you start machining. It is also useful when selecting holes directly, executing holes separately, and skipping holes.

To access the Circle Pattern Table, or Linear Pattern Table, press the **FEATURES** soft key. The **CIRCLE PATTERN**, and **LINEAR PATTERN** soft keys are shown. Pressing one of these soft keys will open the corresponding hole pattern table.

Soft key Functions

The following soft keys are available while in the Circle, and Linear Pattern Milling feature.

Function	Soft key
Press to access the FEATURES soft keys.	
Press to access the circle pattern table.	
Press to access the linear pattern table.	
Press to create a new circle, or linear pattern.	
Press to edit an existing pattern.	
Press CLEAR to delete an existing pattern.	
Press RUN to execute a pattern.	
Press HELP for additional information about the pattern.	

Circle, and Linear Pattern Table Entry

The Circle, or Linear Pattern Table is used to define up to 10 different circle hole patterns (Full, or Segment), and 10 different linear hole patterns (Array, or Frame).

- ▶ Access the Pattern Table from the DRO soft key menu by pressing the FEATURES soft key.
- ▶ Press the CIRCLE PATTERN, or LINEAR PATTERN soft key.
 - The Pattern Table will show all of the patterns that have been previously defined.
- ▶ Press the UP/DOWN arrow keys to define a new pattern, or edit an existing pattern.
- ▶ Highlight the table entry, press the NEW, or EDIT soft key (or press ENTER).
 - A Circle, or Linear Pattern form is opened.
- ▶ Enter the information to define the pattern, then press ENTER.
 - The pattern will be entered into the appropriate table. It can now be modified, executed, or referred to from a program.
- ▶ To delete a pattern from the table press the CLEAR soft key. Press YES to confirm deletion from the table.



The pattern table, and its entries are saved to memory. They will stay in memory until deleted, or changed, and are not affected by a power cycle.

D:0 | T:1 | F: 0.0 | 0:00 | INCH | ABS | |

CIRCLE PATTERN TABLE			
1	FULL	4 (2.0000,1.5000)	5.0000 25.00
2	SEGMENT	3 (1.0000,1.0000)	1.0000 20
3	FULL	8 (10.0000,0.0000)	2.5000 0.0
4			
5			
6			
7			
8			

EDIT CLEAR RUN HELP

D:0 | T:1 | F: 0.0 | 0:00 | INCH | ABS | |

LINEAR PATTERN			
1	ARRAY	(0.0000,0.0000)	5 1.0000
2	FRAME	(1.2500,0.0000)	4 1.5000
3			
4			
5			
6			
7			
8			

NEW HELP

Circle, and Linear Pattern Soft keys

The following additional soft keys are available while in the Circle, and Linear Pattern Milling feature.

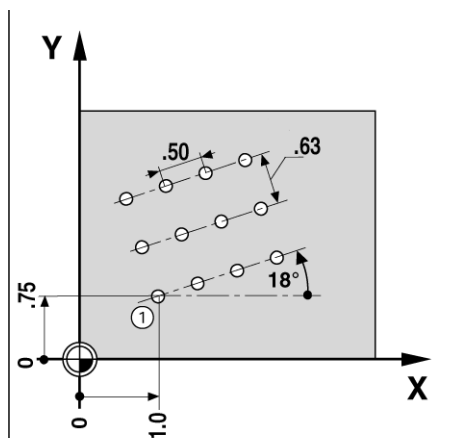
Function	Soft key
Press to execute the circle or linear pattern.	<div style="border: 1px solid black; padding: 5px; display: inline-block;">RUN</div>
Press to use the current absolute position.	<div style="border: 1px solid black; border-radius: 10px; padding: 5px; display: inline-block;">TEACH</div>

Circle Pattern Information Required

- Type: type of pattern, FULL, or SEGMENT.
- Holes: number of holes in the pattern.
- Center: location of the center of the hole pattern.
- Radius: radius of the pattern.
- Start Angle: angle between X axis and the first hole.
- End Angle: angle between X axis and the last hole.
- Depth: the target depth for drilling in the tool axis.

Linear Pattern Information Required




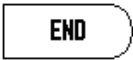
- Type: type of pattern, ARRAY or FRAME.
- First Hole: location of the first hole of the pattern.
- Holes per Row: number of holes in each row of a pattern.
- Angle: the angle, or rotation of the pattern.
- Depth: the target depth for drilling in the tool axis.
- Number of Rows: the number of rows in the pattern.
- Row Spacing: the spacing between each row of the pattern.



Circle, or Linear Pattern Execution

- ▶ To execute a circle, or linear pattern, highlight the pattern to be executed in the table.
- ▶ Press the RUN soft key.
 - The W1000 then calculates the positions of the holes. It can also provide a graphical view of the hole pattern.

The following soft keys are available while running a Circle, or Linear Pattern

Function	Soft key
Press to select the graphic view of the pattern. Available in both incremental DRO, or absolute DRO.	
Press to select the previous hole in the pattern.	
Press to select the next hole in the pattern.	
Press to end execution of the pattern.	

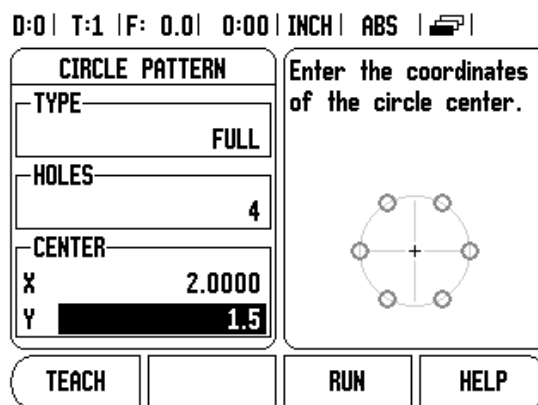


Press the VIEW soft key to toggle the views between distance-to-go positions, graphic view, and absolute positions.

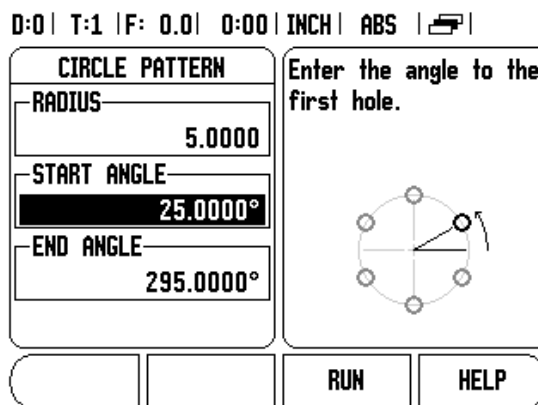
Example: Enter data and execute a circle pattern.

1st step: Enter data


- ▶ Press FEATURES soft key.
- ▶ Press CIRCLE PATTERN soft key.
- ▶ Press UP/DOWN hard keys to select pattern 1.
- ▶ Press ENTER hard key.
- ▶ Press FULL/SEGMENT soft key until FULL is selected.
- ▶ Press DOWN ARROW hard key to move to the next field.
- ▶ Enter 4 for the number of holes.
- ▶ Enter X= 2.0", Y= 1.5" for the circle pattern center location.



- ▶ Enter 5" for the radius of the circle pattern.
- ▶ Enter 25° for the start angle.
- ▶ The end angle is 295°, and can not be changed because the pattern is full.
- ▶ Enter depth of -.25".
 - The depth of the hole is optional, and may be left blank.
- ▶ Press ENTER



The Circle Pattern table now shows the pattern that was just defined as pattern 1.

D:0 | T:1 | F: 0.0 | 0:00 | INCH | ABS | 

CIRCLE PATTERN TABLE			
1	FULL	4 (2.0000,1.5000)	5.0000 25.00
2			
3			
4			
5			
6			
7			
8			

EDIT CLEAR RUN HELP

2nd step: Execute circle pattern

- ▶ Press the RUN soft key.
 - The distance-to-go view is displayed.
- ▶ Move to the hole, move the X, and Y axis until their display values show 0.0.
- ▶ Drill (Z depth): If a depth was entered into the pattern, move Z until it's display value shows 0.0. Otherwise, drill to the desired depth.
- ▶ Press the NEXT HOLE soft key.
- ▶ Continue to drill the remaining holes in the same manner.

When the pattern is complete, press the END soft key.

D:0 | T:1 | F: 0.0 | 0:00 | INCH | INC | H:1/4

-6.5316	X
-2.1130	Y
0.2500	Z
0.0000	W

VIEW PREVIOUS HOLE NEXT HOLE END

Incline, and Arc Milling








The incline and arc milling features provide ways to machine a flat diagonal surface (incline milling) or a rounded surface (arc milling) using a manual machine.

The W1000 provides storage of up to 10 user definable Incline Mill features, and 10 Arc Mill features. Once the features are defined, they are stored in memory and can be recalled at anytime. They can be executed from the DRO, or from the operating program. All defined features will stay in memory until removed, regardless if the power has been cycled.

To access the Incline or Arc Mill Table, press the FEATURE soft key, then press the INCLINE MILL, or ARC MILL soft key to open the corresponding mill table.

Soft Key Functions

The following soft keys are available while in the Milling Feature table.

Function	Soft key
Press to access the Incline Mill Table.	
Press to access the Arc Mill Table.	
Press to create a new milling feature.	
Press to edit and existing milling feature.	
Press to delete and existing milling feature.	
Press to execute the milling feature.	
Press for additional information about the milling feature.	

Incline, and Arc Milling Table Entry

The Incline or Arc Mill Table is used to define the milling features. To access either Mill Table:

- ▶ Press the FEATURES soft key.
- ▶ Press the INCLINE MILL, or ARC MILL soft key.

To define a new milling feature:

- ▶ Press the UP/DOWN ARROW hard keys to highlight a blank feature.
- ▶ Press the NEW soft key, or press ENTER.

To edit an existing feature:


- ▶ Press the UP/DOWN ARROW hard keys to highlight the table entry.
- ▶ Press the EDIT, soft key, or press ENTER.

To execute a milling feature:

- ▶ Highlight the table entry, and press the RUN soft key.
 - See “Incline, & Arc Mill Execution” for more information.


To delete an existing feature:

- ▶ Highlight the table entry.
- ▶ Press the CLEAR soft key.
- ▶ Press YES to confirm deletion.

D:0 | T:1 | F: 0.0 | 0:00 | INCH | ABS |  |

INCLINE MILL TABLE			
1	XY	(0.0000,0.0000)	(-5.0000,5.0000) 0.5000
2			
3			
4			
5			
6			
7			
8			

EDIT CLEAR RUN HELP

D:0 | T:1 | F: 0.0 | 0:00 | INCH | ABS |  |

ARC MILL TABLE			
1	XZ	(0.0000,0.0000)	(-2.5000,0.0000) 0.5000
2			
3			
4			
5			
6			
7			
8			

NEW HELP

Incline, and Arc Milling Table Entry

The following soft keys are available while in the entry form.

Function	Soft key
Press to select a plane ([XY], [YZ], or [XZ])	PLANE [XY]
Press to execute the milling operation	RUN
Press to use the current absolute position	TEACH

The Incline Milling form is used to specify the flat surface to be milled. The data is entered through the "Incline Mill Table". Selecting an incline mill entry, and then pressing the NEW, or EDIT soft key, (or ENTER), will allow the feature to be defined. The following listed fields require data input.

- **Plane:** Select the plane by pressing the PLANE soft key. The current selection is shown on the soft key and in the plane field. The graphic in the message box aids in selecting the correct plane.
- **Start Point:** Enter the coordinates of the start point, or press TEACH to set the coordinate to the current position.
- **End Point:** Enter the coordinates of the end point, or press teach to set the coordinate to current position.
- **Step:** Enter the step size. When milling, this is the distance between each pass or each step along the line.



The Step size is optional. If the value is zero, the operator decides at run-time how far to move between each step.

Press the ENTER key to close the form and save the feature to the table. Press the RUN soft key to execute the surface milling operation. Press C hard key to exit the form without saving the feature.

D:0 | T:1 | F: 0.0 | 0:00 | INCH | ABS |

INCLINE MILL	
PLANE	XY
START POINT	
X	0.0000
Y	0.0000
PLANE [XY]	RUN HELP

D:0 | T:1 | F: 0.0 | 0:00 | INCH | ABS |

INCLINE MILL	
END POINT	
X	5.0000
Y	5.0000
STEP	0.5000
TEACH	RUN HELP

Arc Milling

The Arc Milling form is used to specify the curved surface to be milled. The data is entered through the "Arc Mill Table". Selecting an arc mill entry, and then pressing the NEW, or EDIT soft key, (or ENTER), will allow the feature to be defined.

The following listed fields require data input.

- Plane Selection:** Select the plane by pressing the PLANE soft key. The current selection is shown on the soft key, and in the plane field. The graphic in the message box aids in selecting the correct plane.
- Center Point:** Enter the coordinates of the arc's center point, or press TEACH to set the coordinate to the current position.
- Start Point:** Enter the coordinates of the start point, or press TEACH to set the coordinate to current position.
- End Point:** Enter the coordinates of the end point, or press TEACH to set the coordinate to current position.
- Step:** Enter the step size. When milling, this is the distance along the circumference of the arc between each pass, or step along the arc's contour.



The Step size is optional. If the value is zero, the operator decides at run-time how far to move between each step.

D:0 | T:1 | F: 0.0 | 0:00 | INCH | ABS |

MILL ARC	
PLANE	XZ
CENTER POINT	
X	0.0000
Z	0.0000
PLANE [XZ]	
RUN	HELP

Press PLANE to select the plane.

D:0 | T:1 | F: 0.0 | 0:00 | INCH | ABS |

MILL ARC	
START POINT	
X	-2.5000
Z	0.0000
END POINT	
X	0.0000
Z	-2.5000
TEACH	
RUN	HELP

Enter the incline mill arc FROM point.

Press the ENTER key to close the form and save the feature to the table. Press the RUN soft key to execute the surface milling operation. Press C hard key to exit the form without saving the feature.

Incline, and Arc Mill Execution

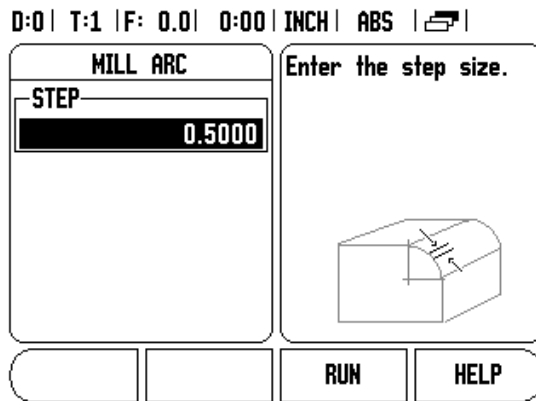
- ▶ To execute a milling operation, select the feature from the table to open the entry form.
- ▶ Press the RUN soft key.
 - The screen switches to the incremental DRO view, showing the incremental distance from the start point.

The following soft keys are available while running an Incline, or Arc Milling feature.

Function	Soft key
Press to select the incremental DRO, contour view of feature, or absolute DRO	VIEW
Press to return to the previous pass.	PREVIOUS PASS
Press to advance to the next pass.	NEXT PASS
Press to end execution of the milling operation.	END

The tool radius compensation is applied based on the radius of the current tool. If the plane selection involves the tool axis, the tool tip is assumed to have a ball end.

- ▶ Move to the start point and make a plunge cut or the first pass across the surface.
- ▶ Press the NEXT PASS, soft key to continue with the next step along the contour.
 - The incremental display shows the distance from the next pass along the line, or arc's contour.



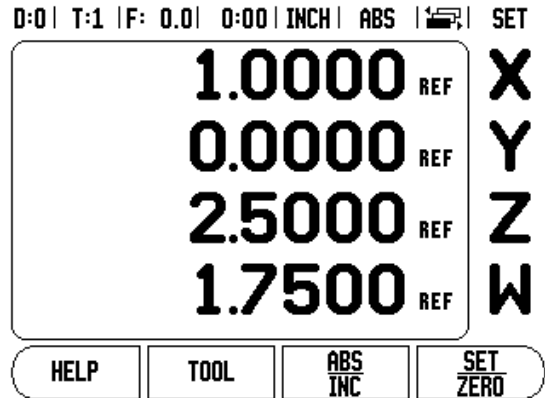
- ▶ To follow the contour, move the two axes in small steps, keeping the **X**, and **Y** positions as close to zero (0.0) as possible.
 - When no step size is specified, the incremental display always shows the distance from the closest point on the arc.
- ▶ Press the VIEW soft key to toggle through the three available views (incremental DRO, contour, and absolute DRO).
 - The contour view shows the position of the tool relative to the milling surface. When the crosshair representing the tool is on the line representing the surface, the tool is in position. The tool crosshair remains fixed in the center of the graph. As the table is moved, the surface line moves.
- ▶ Press END to exit the milling operation.



The tool offset direction (R+ or R-) is applied based on the tool position. The operator must approach the contour surface from the appropriate direction for tool compensation to be correct.

Z/W Coupling (4 axes Milling)

The W1000 Milling application provides a quick method for coupling the Z and W axes positions on a 4 axis system. The display can be coupled in either the Z or W displays.



Enabling Z/W Coupling

To couple the Z and W axes and have the result displayed on the Z display, press and hold the Z key approximately 2 seconds. The sum of the Z/W positions will be displayed on the Z display and the W display will be blanked.

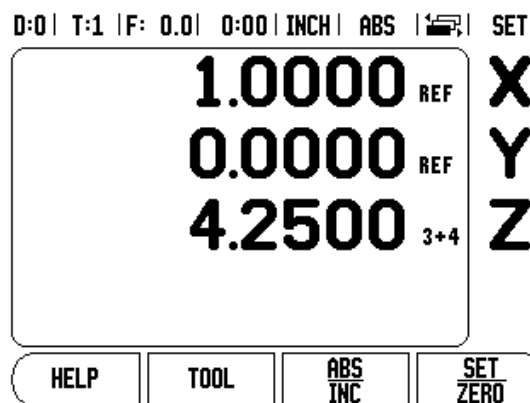
To couple the Z and W axes and have the result displayed on the W display, press and hold the W key for approximately 2 seconds. The sum of the Z/W positions will be displayed on the W display and the Z display will be blanked. The coupling is preserved in between power cycles.

Moving either Z or W inputs will update the coupled Z/W position.

When a position is coupled, the reference mark for both encoders must be found in order to recall the previous datum.

Disabling Z/W Coupling

To disable Z/W Coupling, press the axis key of the display that is blank. The individual Z and W display positions will be restored.



I - 3 Turning Specific Operations


This section discusses operations specific to turning applications only.

Tool Display Icon

The Ø icon is used to indicate that the displayed value is a diameter value. No icon visible indicates that the display is a radius value.

Tool Table

The W1000 can store the dimensional offsets for up to 16 tools (see sample screen). When you change a workpiece and establish a new datum, all tools are automatically referenced from the new datum.

D:0 | T:1 | F: 0.0 | 0:00 | INCH | ABS | 

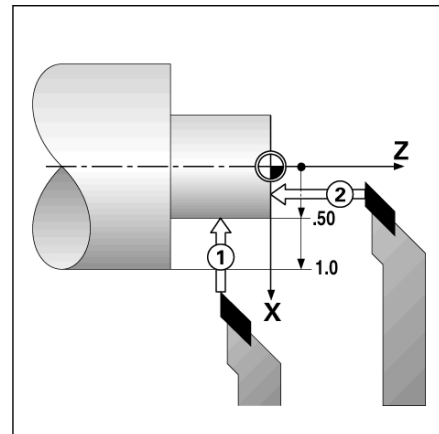
TOOL TABLE (X/Z ₀ /Z/Y)	
1	1.5200Ø
2	2.4500Ø
3	
4	

Setting Tool Offsets Example 1: Using TOOL/SET

Before you can use a tool, you must enter its offset (the cutting edge position). Tool offsets can be set using the TOOL/SET or LOCK AXIS feature.

The TOOL/SET operation can be used to set a tool's offset using a tool when the diameter of the workpiece is known.

- ▶ Touch the known diameter in the X axis (1).
- ▶ Press the TOOL soft key.
- ▶ Scroll to the desired tool.



- ▶ Press the ENTER key.
- ▶ Select the axis (X) key.
- ▶ Enter the position of the tool tip, for example, X= .100. Remember to ensure the W1000 is in diameter display mode (Ø) if the input is a diameter value.
- ▶ Touch the workpiece face with the tool.
- ▶ Cursor to the Z axis (2), then set the position display for the tool tip to zero, Z=0.
- ▶ Press ENTER.

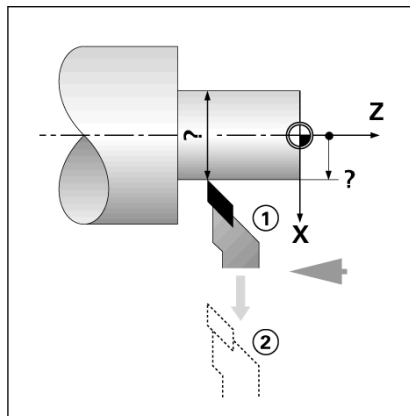
Setting Tool Offsets Example 2: Using LOCK AXIS Function

The LOCK AXIS function can be used to set a tool's offset when a tool is under load and the diameter of the workpiece is not known.

The LOCK AXIS function is useful when determining tool data by touching the workpiece. To avoid losing the position value when the tool is retracted to measure the workpiece, this value can be stored by pressing LOCK AXIS.

To use the LOCK AXIS function:

- ▶ Press the TOOL soft key.
- ▶ Select tool and press ENTER.
- ▶ Press the X axis key.
- ▶ Turn a diameter in the X axis.
- ▶ Press the LOCK AXIS soft key while the tool is still cutting.
- ▶ Retract from the current position.
- ▶ Turn the spindle off and measure the workpiece diameter.
- ▶ Enter the measured diameter or radius, and press ENTER.
Remember to ensure the W1000 is in diameter display mode Ø if you input a diameter value.



Calling a Tool from the Tool Table

- ▶ To call a tool, press the TOOL soft key.
- ▶ Use the Up/Down arrow keys to cursor through the selection of tools (1-16). Highlight the tool you want.
- ▶ Verify the proper tool has been called, and press either the use TOOL soft key, or the C key to exit.

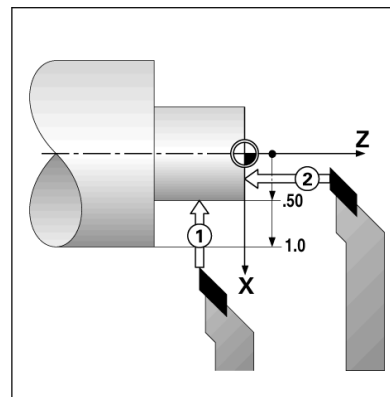
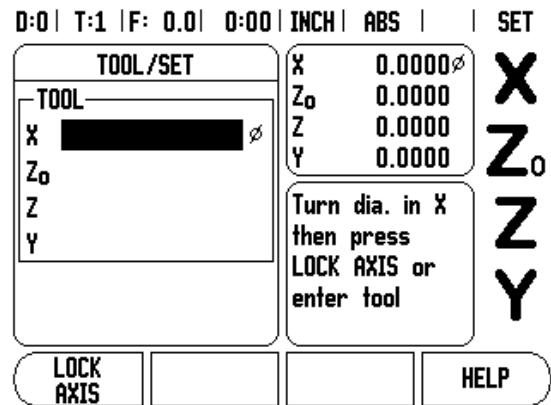
Datum Setting

See "Datum Soft Key" on page 14 for basic information. Datum settings define the relationships between the axis positions and the display values. For most lathe operations there is only one X-axis datum, the center of the chuck, but it may be helpful to define additional datums for the Z-axis. The table can hold up to 10 datum points. The recommended way to set datum points is to touch a workpiece at a known diameter, or location, then enter that dimension as the value that the display should be showing.

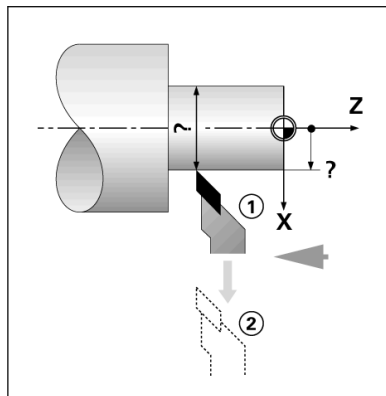
Example: Setting a workpiece datum

Preparation:

- ▶ Call the tool data by selecting the tool which you are using to touch the workpiece.
- ▶ Press the DATUM soft key. The cursor will be in the DATUM NUMBER field.
- ▶ Enter the datum number, and press the DOWN arrow key to go to the X-axis field.
- ▶ Touch the workpiece at point 1.
- ▶ Enter the radius or diameter of the workpiece at that point. Remember to ensure the W1000 is in diameter display mode \emptyset if you input a diameter value.



- ▶ Press the DOWN arrow key to advance to the Z axis.
- ▶ Touch the workpiece surface at point 2.
- ▶ Enter the position of the tool tip (Z= 0) for the Z coordinate of the datum.
- ▶ Press ENTER.



Setting Datums using LOCK AXIS Function

The LOCK AXIS function is useful for setting a datum when a tool is under load and the diameter of the workpiece is not known.

To use the LOCK AXIS function:

- ▶ Press the DATUM soft key. The cursor will be in the DATUM NUMBER field.
- ▶ Enter the datum number and press the DOWN arrow key to go to the X axis field.
- ▶ Turn a diameter in the X axis.
- ▶ Press the LOCK AXIS soft key while the tool is still cutting.
- ▶ Retract from the current position.
- ▶ Turn the spindle off and measure the workpiece diameter.
- ▶ Enter the measured diameter, for example, 1.5" and press ENTER.

D:2 | T:1 | F: 0.0 | 0:00 | INCH | ABS | | SET

SET DATUM		X	2.1842 ϕ	X Z₀ Z Y
DATUM NUMBER		Z ₀	0.0000	
2		Z	0.0000	
DATUM		Y	0.0000	
X	ϕ	Turn dia. in X then press LOCK AXIS or enter tool		
Z ₀				
Z				
Y				
LOCK AXIS				HELP

Taper Calculator Soft Key

Use the taper calculator to calculate taper angle. Refer to screen examples on the right.

You can calculate tapers either by entering dimensions from a print, or by touching a tapered workpiece with a tool or indicator.

Entry values:

For the taper ratio, calculation requires:

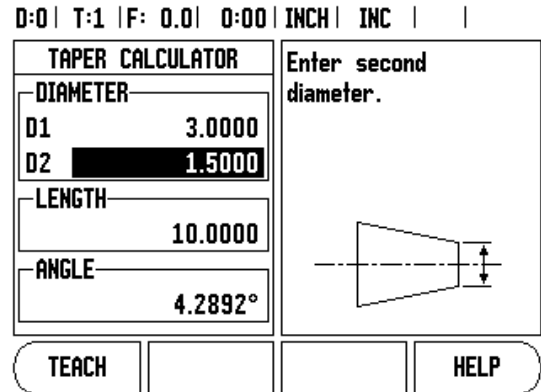
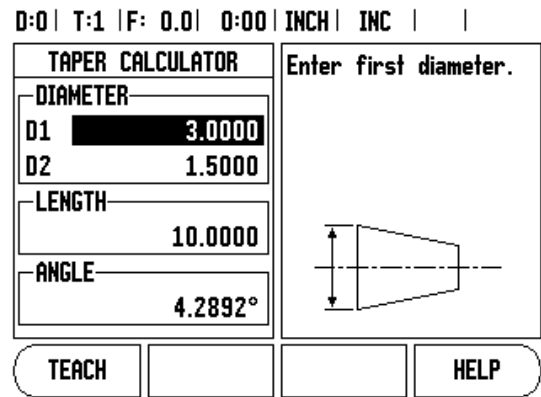
- Length of the taper
- Change in the radius of the taper.

For taper calculation using both diameters (D1, D2) and length requires:

- Starting diameter
- End diameter
- Length of the taper

Using Taper Calculator

- ▶ Press the CALC soft key: The soft key selection now changes to include the taper calculator functions.
- ▶ To calculate the taper angle using two diameters, and length between, press the taper: di/D2/l soft keys.
- ▶ First taper point, diameter 1, either enter a point using the numeric keys and press enter, or touch the tool to one point and press POINT.
- ▶ Repeat this for the diameter 2 field. When using the note key, the taper angle is automatically calculated. When entering data numerically, enter data into the length field, and press ENTER: The taper angle will appear in the angle field.
- ▶ To calculate angles using the ratio of the diameter, change to length, and press the taper: RATIO soft key.
- ▶ Using the numeric keys, enter data into the ENTRY 1, and ENTRY 2 fields.
- ▶ Press ENTER after each selection: The calculated ratio, and the angle will appear in their respective fields.



Presets

The functionality has been explained previously in this manual (see "Presets" on page 17). The explanation, and examples on those pages are based on a mill application. The basics of those explanations are the same for turning applications with two exceptions; Tool Diameter Offsets (R+/-), and Radius vs. Diameter inputs.

Tool diameter offsets have no applications with turning tools, so this functionality is not available while doing turning presets.

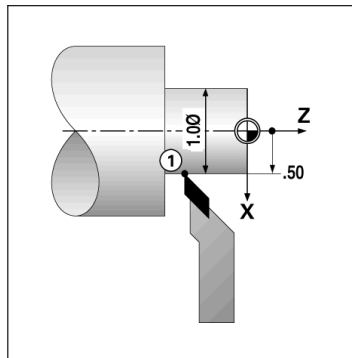
Input values can be either radius or diameter values. It is important to be sure the units you are entering for the preset agree with the state that the display is currently using. A diameter value is shown with a \emptyset symbol. The state of the display can be changed using the RAD/DIA soft key (available in both operating modes).

Radius/Diameter Soft Key

Drawings for lathe parts usually give diameter values. W1000 can display either the radius or the diameter for you. When the diameter is being displayed, the diameter symbol \emptyset is shown next to the position value.

Example:

- Radius display, position 1, X = .50
- Diameter display, position 1, X = \emptyset 1.0
- ▶ Press the RAD/DIA soft key to switch between radius display, and diameter display.



Vectoring

Vectoring breaks down the movement of the compound axis into the crossfeed or longitudinal axes. If you are turning threads, for example, vectoring lets you see the diameter of the thread in the X-axis display, even though you are moving the cutting tool with the compound axis handwheel. With vectoring enabled, you can preset the desired radius or diameter in the X-axis, so that you can “machine to zero”.



When vectoring is used, the top slide (compound) axis encoder must be assigned to the bottom display axis. The crossfeed component of movement of the axis will then be shown in the top display axis. The longitudinal component of movement of the axis will be shown in the middle display axis.

- ▶ Select VECTORING from Job Setup.
- ▶ Press the ON soft key to enable the vectoring feature.
- ▶ Arrow down to the Angle field to enter the angle between the longitudinal slide and top slide with 0° indicating the top slide is moving parallel to the longitudinal slide.
- ▶ Press ENTER.

Z Coupling

The W1000 Turning application provides a quick method for coupling the Z₀ and Z axes positions on a 3, or 4 axis system. The display can be coupled in either the Z₀ and Z displays.

D:0 | T:1 | F: 0.0 | 0:00 | INCH | ABS | | |

VECTORING		Set the angle of the compound axis.
STATUS	ON	
ANGLE	30.0000°	
<input type="button" value="HELP"/>		

D:0 | T:1 | F: 0.0 | 0:00 | INCH | ABS | | SET

1.0000	∅ REF	X
2.5000	REF	Z₀
1.7500	REF	Z
0.0000	REF	Y

HELP	TOOL	ABS INC	SET ZERO
------	------	------------	-------------

Enabling Z₀ and Z Coupling

To couple the Z₀ and Z axes and have the result displayed on the Z₀ display, press and hold the Z₀ key approximately 2 seconds. The sum of the Z positions will be displayed on the Z₀ display and the Z display will be blanked.

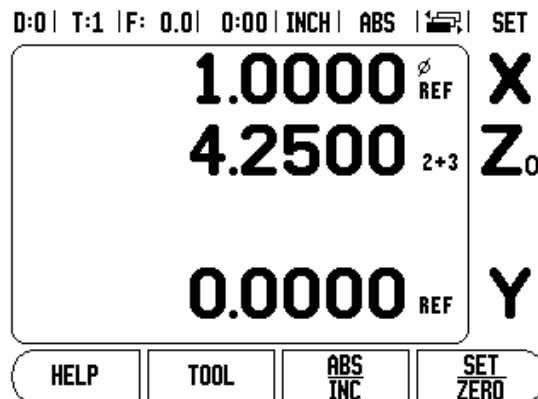
To couple the Z₀ and Z axes and have the result displayed on the Z display, press and hold the Z key for approximately 2 seconds. The sum of the Z positions will be displayed on the Z display and the Z₀ display will be blanked. The coupling is preserved in between power cycles.

Moving either Z₀ or Z inputs will update the coupled Z position.

When a position is coupled, the reference mark for both encoders must be found in order to recall the previous datum.

Disabling Z₀ and Z Coupling

To disable Z Coupling, press the axis key of the display that is blank. The individual Z₀ and Z display positions will be restored.



I - 4 Programming W1000

The basic machining operations available in the DRO mode (i.e. tool selection, preset, hole patterns) can also be used to create a program. A program is a sequence of one or more machining operations. Programs can be executed a number of times and saved for later use.

Each machining operation is a separate step with the program. A program can have up to 250 steps. The program listing shows the step numbers, and the associated machining operations.

Programs can be saved to internal storage, which isn't lost when power is off. The readout can hold up to 8 programs in internal storage. Programs can also be saved to external storage on a PC using the export and import functions.

To access the Program Mode, from the DRO mode, press the PROGRAM soft key. A listing of the current program loaded (or a blank listing) is displayed. Also displayed are the soft keys VIEW, FEATURES, RUN, HELP, PROGRAM FUNCTIONS, and STEP FUNCTIONS.

Press the LEFT, or RIGHT ARROW keys to view more soft key choices.

D:0 | T:1 | F: 0.0 | 0:00 | INCH | ABS |

CURRENT PROGRAM		X	0.0000
001	DATUM 0	Y	0.0000
002	TOOL 1	Z	0.0000
003	POSITION	W	0.0000
004	POSITION	Press ENTER to edit step.	
005	POSITION		
006	POSITION		
007	POSITION		
008	POSITION		

VIEW FEATURES RUN HELP







D:0 | T:1 | F: 0.0 | 0:00 | INCH | ABS |

CURRENT PROGRAM		X	0.0000
001	DATUM 0	Y	0.0000
002	TOOL 1	Z	0.0000
003	POSITION	W	0.0000
004	POSITION	Press ENTER to edit step.	
005	POSITION		
006	POSITION		
007	POSITION		
008	POSITION		

PROGRAM FUNCTIONS STEP FUNCTIONS HELP

Program Mode Soft key functions

The following soft keys are available while in the Program Mode feature.

Function	Soft key
Press to toggle between the current program listing with DRO (ABS) positions and a graphical view of the programmed part (milling only).	
Press to show, and select the available programming features: TOOL, DATUM, PRESET, and for milling: POSITION, CIRCLE PATTERN, LINEAR PATTERN, INCLINE MILL, and ARC MILL.	
Press to execute the program from the current highlighted step.	
Press to display help relating to the program mode.	
Press to show the available program functions: LOAD, SAVE, DELETE, CLEAR, IMPORT, and EXPORT.	
Press to show the available step functions: CLEAR STEP, and EXPLODE STEP (milling only).	

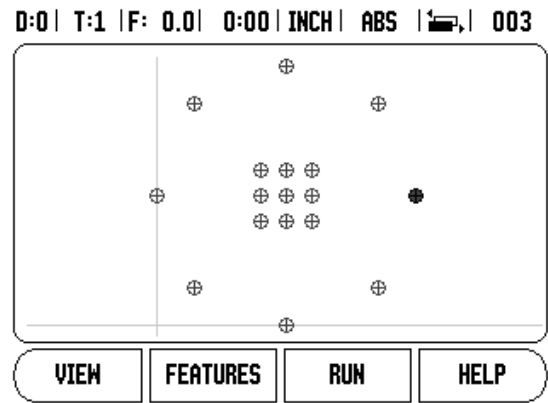
View Soft Key

The VIEW soft key is used to toggle between the program listing with the DRO (ABS) positions, and graphical view of the part program.

When Program Mode is selected, the view defaults to the program listing view.





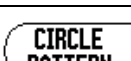


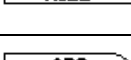
Press the **VIEW** soft key to show a graphical view of the part program. The datum, position, circle pattern, and linear pattern steps have a graphical view. The features of the part are scaled to the size of the window.

- The datum is drawn as a vertical and horizontal line representing the (0,0) location.
- The position and holes of a hole pattern are drawn as circles. The diameter of the programmed tool is used to draw the holes.



Feature Soft Key functions

To access the available features, press **PROGRAM**, then the **FEATURES** soft key. The following soft keys will be available. Use the **LEFT**, and **RIGHT** arrow keys to toggle through the menu. Note that in program mode the **C** key will clear the entry

Function	Soft key
Press to enter a Tool step into the program. See "Tool Soft Key" on page 12.	
Press to enter a Datum step. See "Datum Soft Key" on page 14.	
Press to enter a Preset step into the program. See "Presets" on page 17.	
Press to enter a Position step (mill only).	
Press to enter a Circle Pattern step (mill only). See "Circle, and Linear Patterns" on page 20.	
Press to enter a Linear Pattern step (mill only). See "Circle, and Linear Patterns" on page 20.	
Press to enter an Incline Mill step (mill only). See "Incline, and Arc Milling Table Entry" on page 28.	
Press to enter an Arc Mill step (mill only). See "Arc Milling" on page 30	

Tool Soft Key

The Tool step is used to select the tool (from the Tool Table) that will be used by the subsequent steps in the program. Open the Tool Table by pressing the TOOL soft key. Select the desired tool, then press the USE TOOL soft key. The step that was highlighted in the program will become a Tool step.

Datum Soft Key

The datum step is used to select the datum to be used by subsequent steps in the program. Open the Datum form by pressing the DATUM soft key. Enter the desired datum number (0-9), then press ENTER. The step that was highlighted in the program will become a Datum step.

Preset Soft Key

The preset step is used to enter the nominal (target) position to move to from within a program. Open the Preset form by pressing the PRESET soft key. Select the desired information, then press ENTER. The step that was highlighted in the program will become a Preset step.

Position Soft Key (mill only)

The Position step is used to enter the nominal (target) position to move to "for the machine and tool axes" (X, Y, & Z) from within a program. Open the Position form by pressing the POSITION soft key. Enter the nominal positions for the machine axes, and the depth (optional) for the tool axis, then press ENTER. The depth is optional, and can be left blank. The step that was highlighted in the program will become a Position step.

Circle Pattern Soft Key (mill only)

The CIRCLE PATTERN soft key is used to specify the parameters of a circle pattern from within a program. Open the Circle Pattern table by pressing the CIRCLE PATTERN soft key. Use the UP, and DOWN arrow keys to select the desired pattern, then press USE. The step that was highlighted in the program will become a Circle Pattern step.

To modify the selected pattern, press ENTER, or the EDIT soft key. In the form; enter the new pattern parameters, then press ENTER.

Linear Pattern Soft Key (mill only)

The LINEAR PATTERN soft key is used to specify the parameters of a linear pattern from within a program. Open the Linear Pattern table by pressing the LINEAR PATTERN soft key. Use the UP, and DOWN arrow keys to select the desired pattern, then press USE. The step that was highlighted in the program will become a Linear Pattern step.

To modify the selected pattern, press ENTER, or the EDIT soft key. In the form; enter the new pattern parameters, then press ENTER.

Incline Mill (mill only)

The INCLINE MILL soft key is used to provide a way to machine a flat diagonal surface from within a program. Open the Incline Mill Table by pressing the INCLINE MILL soft key. Use the UP, and DOWN arrow keys to select the desired incline mill feature, then press USE. The step that was highlighted in the program will become an Incline Mill step.

To modify the selected pattern, press ENTER, or the EDIT soft key. In the form; enter the new pattern parameters, then press ENTER.







Arc Mill (mill only)

The ARC MILL soft key is used to provide a way to machine a rounded surface from within a program. Open the Arc Mill Table by pressing the ARC MILL soft key. Use the UP, and DOWN arrow keys to select the desired arc mill feature, then press USE. The step that was highlighted in the program will become an Arc Mill step.

To modify the selected pattern, press ENTER, or the EDIT soft key. In the form; enter the new pattern parameters, then press ENTER.



Program Function Soft keys

The following Program Function soft keys are available while in Program Mode

Function	Soft key
Press to load a previously saved program into current program.	
Press to save and name the current program.	
Press to delete a saved program permanently.	
Press to clear current program memory.	
Press to Import a program into current program memory over the RS-232 port from a PC.	
Press to Export the current program over the RS-232 port to a PC.	

Step Function Soft keys

The following Step Function soft keys are available while in Program Mode.

Function	Soft key
Press to clear the highlighted step from current program.	
Press to explode highlighted step into position steps. This applies only to Circle and linear Patterns.	

Editing, and moving through a program

- ▶ Press UP/DOWN arrow keys to move the highlighted cursor to the next or previous step in the program listing.
- ▶ To move to a specific step in the program use the numeric keys and enter the desired step number (i.e. 005).
- ▶ To insert a new step in the program, highlight the step in the program listing, press the FEATURES soft key and select the desired feature.



The highlighted step and all of the following steps are shifted down one step in the program listing, and the new step is inserted at the highlighted step.

- ▶ To edit the step currently highlighted, press the ENTER key.
- ▶ To clear a step, press STEP FUNCTIONS soft key, then press the CLEAR STEP soft key. A YES/NO confirmation is required before the step will be removed from the program listing.



When a step is deleted from the program listing all of the steps following the deleted step are shifted up one step in the program listing

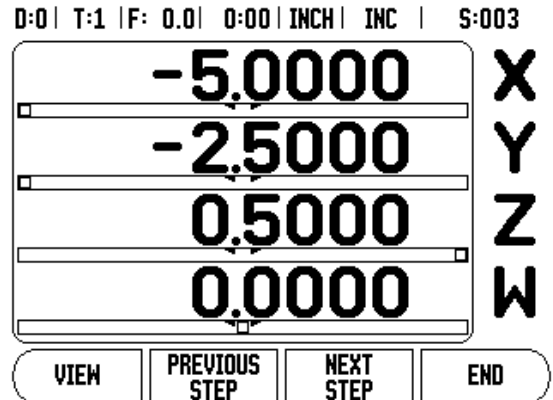
- ▶ To run the current program, press the RUN soft key. The program will be executed from the current step that is highlighted.

I - 5 Executing a Program

To run a program, use the UP/DOWN arrow keys, or use numeric keys to select the program step to begin running. Press RUN. The distance-to-go view is displayed, and the soft keys VIEW, PREVIOUS STEP, NEXT STEP, and END, are shown. The number of the current step being executed is also displayed on the status bar.

Run Soft Keys

Function	Soft key
Press to see the graphical view of the circle or linear pattern that is being executed.	
Press to go to previous step in program	
Press to go to next step in program.	
Press to go to next hole in program.	
Press to go to previous hole in program	
Press to end program execution	



Executing Steps

When a TOOL step is executed, the tool number on the status bar flashes. This indicates that the tool needs to be changed to the tool number shown on the status bar.

When a DATUM step is executed, the datum number on the status bar flashes indicating that the datum is about to be changed and this will be the datum used for the following program operations.

When a POSITION step is executed, the display show the distance-to-go for the machine and tool axes. Move the axes until the positions read 0.0, then press NEXT STEP.

When a PRESET step is executed, the display shows the distance-to-go. Move the axes until the positions read 0.0, then press NEXT STEP.

When a CIRCLE PATTERN, or LINEAR PATTERN step is executed, the display shows the distance-to-go to get to each hole in the pattern. Move the axes until the positions read 0.0, then press NEXT HOLE.

Once the correct operation is performed, press NEXT STEP to get the next step in the program and execute it. Pressing the PREVIOUS STEP will execute the previous step in the program.

If the current step being executed is a circle or linear pattern, the VIEW, PREVIOUS HOLE, and NEXT HOLE soft keys are shown. Pressing NEXT HOLE will display the distance-to-go for the next hole in the pattern. Once all holes in the step have been executed the soft keys become PREVIOUS STEP, and NEXT STEP. Press the VIEW soft key to toggle between the distance-to-go view, and the hole pattern graphic view.

If the current step being executed is for incline milling, or arc milling, the VIEW, PREVIOUS PASS, and NEXT PASS soft keys are shown. The screen switches to the DRO (INC) view, and shows the incremental distance from the start point. After moving to the start point, Press NEXT PASS the incremental display shows the distance from the next pass along the line's (or arc's) contour. When the last pass of the milling operation is executed, the NEXT PASS soft key changes to NEXT STEP.

Pressing END or executing a blank step in the program completes the run mode operation and returns to the Program Mode.

II - 1 Installation Setup

Installation Setup Parameters

To locate the Installation Setup screen from the normal operating display:

- ▶ Press the RIGHT, or LEFT arrow key until the SETUP soft key is displayed
- ▶ Press the SETUP soft key.
 - The Job Setup screen is now displayed, and the INSTALL. SETUP soft key is displayed.
- ▶ Press the INSTALL. SETUP soft key, and enter the passcode.
 - The Installation Setup screen is now displayed.

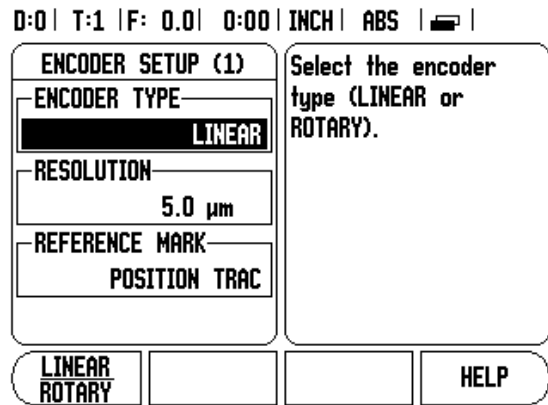
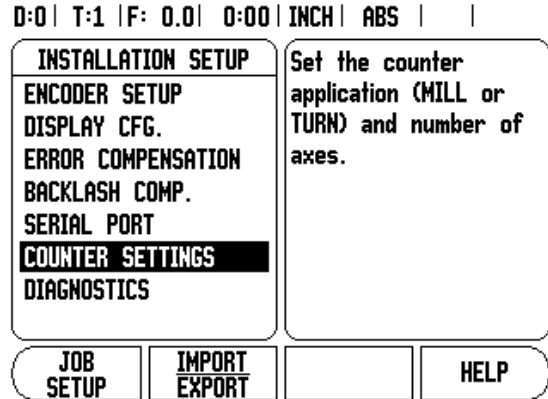
Installation Setup parameters are established during the initial installation, and do not often change. For this reason, the installation setup parameters are protected by the passcode.

See "Readout Parameter Access Code" on page iii

Encoder Setup

The ENCODER SETUP is used to set the encoder: resolution, type (linear, rotary), count direction, and reference mark type.

- ▶ Press the UP, or DOWN arrow keys until ENCODER SETUP is highlighted.
- ▶ Press ENTER.
 - This opens a list of possible encoder inputs.
- ▶ Highlight the encoder to be setup using the UP, or DOWN arrow keys.
- ▶ Press ENTER.
 - Cursor will be in the ENCODER TYPE field.



- ▶ Select the encoder type by pressing the LINEAR/ROTARY soft key.
- ▶ Arrow down to the RESOLUTION field.
- ▶ Press COARSER, or FINER soft keys when using linear encoders to select the encoder's resolution in μm (10, 5, 2, 1, 0.5). The exact resolution can also be typed in using the numeric key pad.
 - Rotary encoders require the number of counts per revolution to be entered using the numeric key pad.
- ▶ Arrow down to the REFERENCE MARK field.
- ▶ Toggle the REF MARK soft key.
 - Select NONE for no reference signal, SINGLE for single reference mark, or POSITION TRAC for an encoder with the Position-Trac™ feature.
- ▶ Arrow down to the COUNT DIRECTION field.
- ▶ In the COUNT DIRECTION field, select the count direction by pressing the POSITIVE/NEGATIVE soft key.
 - When the encoder's count direction matches the operators count direction, select positive. When they do not match, select negative.
- ▶ Arrow down to the ERROR MONITOR field.
- ▶ In the ERROR MONITOR field, select whether the system will monitor, and display encoder errors by selecting ON, or OFF.
 - When an error message occurs, press the C key to remove it.
- ▶ Press ENTER to save the settings entered, and move to the next encoder setup, or press the C key to remove them.
- ▶ To exit and save setting, press ENTER, then JOB SETUP.



The encoder resolution and count direction can also be established by just moving each axis.

Display Configuration

The Display Configuration form is where which axes can be selected for display, and in what order. The following additional setups are also available.

- ▶ Scroll to the desired display, and press ENTER.
- ▶ Press the ON/OFF soft key to turn the display on, or off. Press the LEFT, or RIGHT arrow key to select the axis label.
- ▶ Scroll to the Input field. Press the numeric key associated with the encoder input on the back of the unit.
- ▶ Press the + or - soft keys to couple a second input with the first. The input numbers are displayed next to the axis label indicating that the position is a coupled position (i.e. "2 + 3").
- ▶ Scroll to the Display Resolution field. Press the COARSER, or FINER soft keys to select the display resolution.
- ▶ Scroll to the Angle Display field if the encoder type is set to Rotary. Press the ANGLE soft key to display the position as 0° - 360°, ± 180°, ± infinity, or RPM.

Error Compensation

The distance a cutting tool travels, measured by an encoder, can in certain cases, differ from the actual tool travel. This error can occur due to ball screw pitch error, or deflection and tilting of axes. This error can either be linear, or non-linear. You can determine these errors with a reference measurement system, such as gauge blocks, laser, etc. From an analysis of the error, it can be determined which form of compensation is required, linear, or non-linear error.

The W1000 provides the opportunity to compensate for these errors, and each axis can be programmed separately with the appropriate compensation.



Error compensation is only available when using linear encoders.

Linear Error Compensation

Linear error compensation can be applied, if the results of the comparison with a reference standard show a linear deviation over the whole measuring length. In this case the error can be compensated by the calculation of a single correction factor.

To calculate the linear error compensation use this formula:

Correction factor **LEC = (S – M) x 10⁶ ppm** with:

S measured length with reference standard

M measured length with device at axis

Example:

If the length of the standard you used is 500 mm, and the measured length along the X-axis is 499.95, then the LEC for the X-axis is 100 parts per million (ppm).

LEC = (500 – 499.95) x 10⁶ ppm = 100 ppm (rounded to the nearest whole number).


- ▶ Once determined, the encoder's error information is entered directly. Press the TYPE soft key to select LINEAR compensation.
- ▶ Enter the compensation factor in parts per million (ppm), and press the ENTER key.

Automatic Linear error compensation

The correction factor may be calculated automatically using a standard gauge block. Press the AUTO CALC soft key. In the Standard Field, enter the length of the standard plus any tool offset (tool, or edge finder diameter), which is not required when measuring from the same direction.

To measure the standard using an edge finder, touch one end of the block. The Measured Field shows 0. Move to the second edge, and touch the other end of the gauge block. The measured size, plus any tool offset appears in the field.

To measure the standard using a tool, touch one end of the block, and press TEACH. The Measured Field shows 0. Move to the second edge, and touch the other end of the gauge block. Press TEACH. The measured size, plus any tool offset appears in the field.

D:0 | T:1 | F: 0.0 | 0:00 | INCH | ABS | 

ERROR COMPENSATION	
INPUT 1	0 PPM
INPUT 2	OFF
INPUT 3	OFF
Error compensation for this input is OFF. Press TYPE to select linear or non-linear error compensation.	
TYPE [OFF]	HELP

The Factor Field shows the calculated correction factor based on the measured and standard value.

Press the ENTER key to exit the form and transfer the calculated factor to the error compensation form, or C to exit without transferring the factor.

Non-Linear Error Compensation

Non-linear error compensation should be applied if the results of the comparison with a reference standard show an alternating, or oscillating deviation. The required correction values are calculated, and entered in a table. The W1000 supports up to 200 points per axis. The error value between two entered adjacent correction points is calculated with linear interpolation.



Non-linear error compensation is only available on scales with reference marks. If non-linear error compensation has been defined, no error compensation will be applied until the reference marks have been crossed.

Starting a Non-Linear Error Compensation Table

- ▶ Select Non-linear by pressing the TYPE soft key.
- ▶ To start a new error compensation table, first press the EDIT TABLE soft key.
- ▶ All correction points (up to 200) are equally spaced from the start point. Enter the distance between each of the correction points. Press the DOWN arrow key.
- ▶ Enter the table's start point. The start point is measured from the scale's reference point. If this distance is not known, you can move to the location of the start point, and press TEACH POSITION. Press ENTER.



Pressing ENTER will save the spacing, and start point information.

Configuring the Compensation Table

- ▶ Press the EDIT TABLE soft key to view the table entries.
- ▶ Use the UP, or DOWN arrow keys, or the numeric keys to move the cursor to the correction point to be added or changed. Press ENTER.
- ▶ Enter the known error which exists at this point. Press ENTER.
- ▶ When completed, press C key to exit the table and return to the Error Compensation form.

Reading the Graph

The error compensation table may be viewed in table or graphical formats. The graph shows a plot of a translation error vs. measured value. The graph has a fixed scale. As the cursor is moved through the form, the location of the point on the graph is indicated with a vertical line.

Viewing the Compensation Table

- ▶ Press the EDIT TABLE soft key.
- ▶ To switch between the table and graph views, press the VIEW soft key.
- ▶ Press the UP, or DOWN arrow keys or the numeric keys to move the cursor within the table.

The error compensation table data may be saved to or loaded from a PC via the Serial port.

Automatic Non Linear error compensation

The correction factor may be calculated automatically using a standard gauge block. To select automatic calculation, move the cursor to Spacing, and press ENTER. Press the MANUAL/AUTOMATIC soft key to select automatic which is now displayed in the Spacing Field. Move the cursor to any entry within the table (000-199), and press ENTER.

In the Standard Field, enter the length of the standard plus any tool offset (tool or edge finder diameter), which is not required when measuring from the same direction.

To measure the standard using an edge finder, touch one end of the block. The Measured Field shows 0. Move to the second edge, and touch the other end of the gauge block. The measured size, plus any tool offset appears in the field.

To measure the standard using a tool, touch one end of the block, and press teach. The Measured Field shows 0. Move to the second edge, and touch the other end of the gauge block. Press TEACH. The measured size, plus any tool offset appears in the field.

The Factor Field shows the calculated correction factor based on the measured and standard value.

Press the ENTER key to exit the form and update the table with information on the new interval, or C to exit without transferring the factor.

Exporting the current Compensation Table

- ▶ Press the EDIT TABLE soft key.
- ▶ Press the IMPORT/EXPORT soft key.
- ▶ Press the EXPORT TABLE soft key.

Importing a new Compensation Table

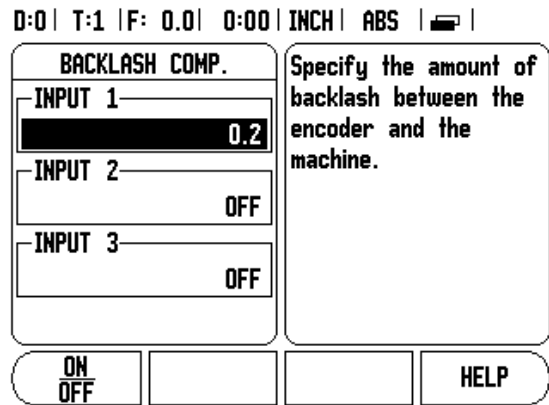
- ▶ Press the EDIT TABLE soft key.
- ▶ Press the IMPORT/EXPORT soft key.
- ▶ Press the IMPORT TABLE soft key.

Backlash Compensation

When using a rotary encoder with a lead screw, a change in direction of the table might cause an error in the displayed position due to clearances within the lead screw assembly. This clearance is referred to as backlash. This error can be compensated for by inputting the amount of backlash within the lead screw into the Backlash Compensation feature. See screen to the right.

If the rotary encoder is ahead of the table (displayed value is greater than the table’s true position), this is called positive backlash, and the value entered should be the positive value of the amount of error.

No Backlash Compensation is 0.000.




Serial Port

A printer or computer may be connected to the serial port. Job, and Installation Setup parameters may be sent to a printer or computer. Remote commands, remote key codes, and Job, and Installation Setup parameters may be received from a computer.

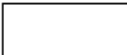
- ▶ The BAUD Field can be set to 300, 600, 1 200, 2 400, 9 600, 19 200 38 400, 57 600, or 115 200 by using the soft keys.
- ▶ The parity can be set to NONE, EVEN, or ODD using the LOWER, or HIGHER soft key.
- ▶ Data Bits in the Format Field can be set to 7, or 8 using the soft key provided.
- ▶ The Stop Bits Field can be set to 1, or 2 using soft key.
- ▶ Line Feed Field can be set to YES if the external device needs a line feed to follow a carriage return.
- ▶ Output tail is the number of carriage returns that will be sent at the end of the measured value output transmission. The output tail is initially 0, and can be set to a positive integer value (0 - 9) by using the numeric hard keys.

The serial port settings will be retained on a power cycle. There is no parameter to enable, or disable the serial port. Refer to Data Interface section for cable connection, and pin assignments.

D:0 | T:1 | F: 0.0 | 0:00 | INCH | ABS |  |

SERIAL PORT	
BAUD	9600
PARITY	NONE
FORMAT	
DATA	8 BITS
STOP	1 BITS

Press the LOWER or HIGHER soft key to match the baud setting of the external device.

LOWER HIGHER  HELP

Counter Settings

The COUNTER SETTINGS feature is the parameter where the operator defines the user application for the readout. The choices are for milling, or turning applications.

A FACTORY DEFAULT soft key appears in the Counter Settings choice of options. When pressed, the configuration parameters (based on either mill, or turn) will be reset to factory defaults. A responses of YES to set parameters to factory default settings, or NO to cancel, and return to previous menu screen is required.


The Number of Axes field sets the number of axes needed. A 1, 2, OR 3 soft key will appear to choose between either 1, 2 or 3 axes.

D:0 | T:1 | F: 0.0 | 0:00 | INCH | ABS | |

COUNTER SETTINGS	
APPLICATION	MILL
NUMBER OF AXES	4
POSITION RECALL	OFF

Set the application to MILL or TURN.

Press FACTORY DEFAULTS to reset all parameters to their factory default settings.

MILL FACTORY  HELP
TURN/GP DEFAULTS

The Position Recall feature, when it is "ON", will store the last position of each axis when power was turned off, and then redisplay that position once power is turned back on.



Note that any movement that occurs while power is off will be lost. Whenever power has been off it is recommended to re-establish workpiece datums using the Reference Mark Evaluation procedure. .

Diagnostics

The DIAGNOSTICS provides access for testing the keypad, and edge finders.

Keypad Test

An image of the keypad provides an indication when a switch is pressed, and released.

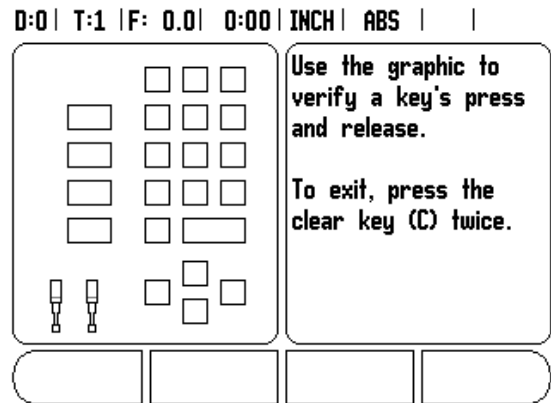
- ▶ Press each hard, and soft key to test. A dot will appear on each key when it has been pressed indicating that it is operating properly.
- ▶ Press the C key two times to exit the keypad test.

Edge Finder Test

- ▶ To test the edge finder, touch the edge finder to a part. An * will appear on the display screen above the corresponding edge finder in use. Grounding type edge finder symbol is to the left, and an electronic edge finder symbol is to the right.

Display Test

- ▶ To test the display, press the ENTER key to cycle through the available colors.



II - 2 RS-232C Serial Communications

Serial port

The RS-232-C/V.24 serial port is located on the rear panel. The following devices can be connected to this port:

- Printer with serial data interface.
- Personal computer with serial data interface.

For operations that support data transfer, an IMPORT/EXPORT soft key is available. See "Import/Export" on page 10.

To export, or import data between the W1000 and a personal computer, the PC can use a common terminal communications software such as Hyperterminal. This software processes the data being sent, or received over the serial cable link. All data transferred between the W1000, and the PC is in ASCII text format.

To export data from the W1000 to a PC, the PC must first be made ready to receive the data to save it to a file. Setup the terminal communication program to capture ASCII text data from the COM port to a file on the PC. After the PC is ready to receive, start the data transfer by pressing the W1000 IMPORT/EXPORT soft key.

To import data into the W1000 from a PC, the W1000 must first be made ready to receive the data.

- ▶ Press the W1000 IMPORT/EXPORT soft key.
- ▶ After the W1000 is ready, setup the terminal communications program on the PC to send the desired file in ASCII text format..



The W1000 does not support communication protocols such as Kermit or Xmodem.

II - 3 Installation and Electrical Connections

Installation

The DRO is mounted to a tilt/swivel feature: See "DRO Dimensions" on page 71.

Electrical requirements

Voltage 100 - 240 Vac

Power 30 VA max.

Frequency 50/60 Hz (+/- 3Hz)

Fuse 630 mA/250 Vac, 5 mm x 20 mm, Slo-Blo (line and neutral fused)

Environmental

Operating temperature 0° to 45°C (32° to 113°F)

Storage temperature -20° to 70°C (-4° to 158°F)

Mechanical weight 2.6 kg (5.2 lb.)

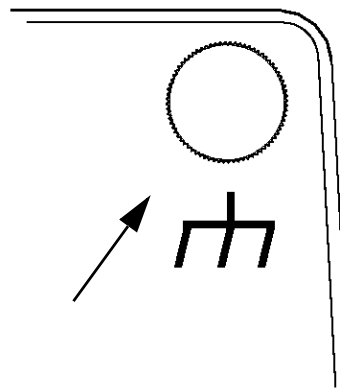
Protective earthing (grounding)



It is necessary to connect the protective conductor terminal on the rear panel to the star point of machine ground as shown in figure to the right.

Preventative maintenance

No special preventative maintenance is necessary. For cleaning, wipe lightly with a dry lint-free cloth.

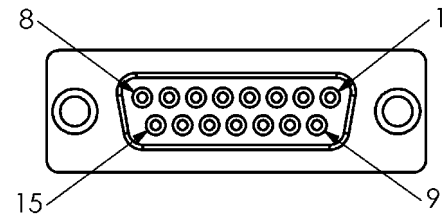


II - 4 I/O Connections

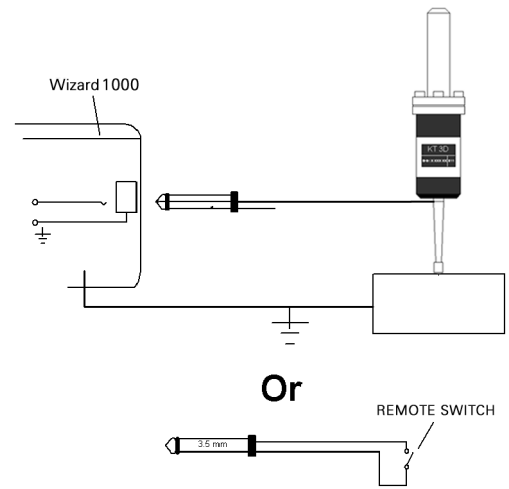
Encoders are plugged into connectors marked Inputs 1, 2, 3, & 4.

Pin layout for Electronic Edge Finder

Pin	Assignment
1	0V (Inner shield)
2	Stand By
3	
4	
5	
6	+5V
7	
8	0V
9	
10	
11	
12	
13	Switch Signal
14	
15	
Housing	External Shield



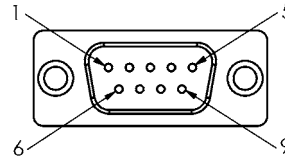
15 Pin receptacle connector for Edge Finder.



Grounding Edge Finder / Remote Switch

Wiring the Serial communication cable

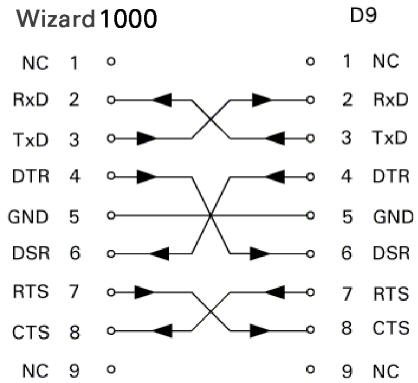
The wiring of the serial communication cable depends on the device being connected (see technical documentation for external device)



Pin layout of RS-232-C / V.24 data interface.

Pin assignment

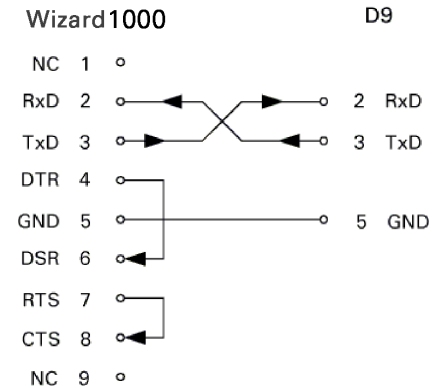
Pin	Assignment	Function
1	No assignment	
3	TXD	- Transmitted Data
2	RXD	- Received data
7	RTS	- Request to send
8	CTS	- Clear to send
6	DSR	- Data set ready
5	SIGNAL GND	- Signal ground
4	DTR	- Data terminal ready
9	No assignment	



Pin connection for serial port with handshaking.

Signal

Signal	Signal Level "1" = "active"	Signal level "0" = "inactive"
TXD, RXD	-3 V to - 15 V	+3 V to + 15 V
RTS, CTS DSR, DTR	+3 V to + 15 V	-3 V to - 15 V



Pin connection for serial port without handshaking.

II - 5 Remote Switch Data Output

The remote switch (pendant or footswitch), or **Ctrl B** (sent over serial interface), will transmit the currently displayed values in either Actual Value or Distance-To-Go mode, whichever is currently visible.

Data output using external signals

Example 1: Linear axis with radius display X = + 41.29 mm

X	=	+	4 1	.	2 9		R	<CR>	<LF>
1	2	3	4	5	6	7	8	9	10

- 1 Coordinate axis
- 2 Equal sign
- 3 +/- sign
- 4 2 to 7 places before the decimal point
- 5 Decimal point
- 6 1 to 6 places after the decimal point
- 7 Unit: blank space for mm, " for inches
- 8 Absolute display:
R for radius, **D** for diameter
 Distance-to-go display:
r for radius, **d** for diameter
- 9 Carriage return
- 10 Blank line (Line Feed)

Example 2: Rotary axis with degrees decimal display
C = + 1260.0000°

C	=	+	1 2 6 0	.	0 0 0 0		W	<CR>	<LF>
1	2	3	4	5	6	7	8	9	10

- 1 Coordinate axis
- 2 Equal sign
- 3 +/- sign
- 4 4 to 8 places before the decimal point
- 5 Decimal point
- 6 0 to 4 places after the decimal point
- 7 Blank space
- 8 **W** for angle (in distance-to-go display: **w**)
- 9 Carriage return
- 10 Blank line (Line Feed)

Example 3: Rotary axis with degrees/minutes/seconds display
C = + 360° 23' 45"

C	=	+	3 6 0	:	2 3	:	4 5		W	<CR>	<LF>
1	2	3	4	5	6	7	8	9	10	1	2

- 1 Coordinate axis
- 2 Equal sign
- 3 +/- sign
- 4 3 to 8 places degrees
- 5 Colon
- 6 0 to 2 places minutes

- 7 Colon
- 8 0 to 2 places seconds
- 9 Blank space
- 10 **W** for angle (in distance-to-go display: **w**)
- 11 Carriage return
- 12 Blank line (Line Feed)

Data output using Edge Finder

In the next three examples, measured value output is started with a **switching signal from the edge finder**. Printing capability can be turned on, or off in the Job Setup parameter Measured Value Output. Information from here is transmitted from the selected axis.

Example 4: Probing function Edge Y = -3674.4498 mm

Y		:	-	3 6 7 4	.	4 4 9 8		R	<CR>	<LF>
1	2	3	4	5	6	7	8	9	10	11

- 1 Coordinate axis
- 2 (2) Blank Spaces
- 3 Colon
- 4 +/- sign, or Blank Space
- 5 2 to 7 places before the decimal point
- 6 Decimal point
- 7 1 to 6 places after the decimal point
- 8 Unit: blank space for mm, " for inches
- 9 **R** for Radius, **D** for Diameter display
- 10 Carriage return
- 11 Blank line (Line Feed)

Example 5: Probing function Centerline

Coordinate of centerline on X axis CLX = + 3476.9963 mm (**C**enter
Line **X** axis)

Distance between the probed edges DST = 2853.0012 mm (**D**istance)

CLX	:	+	3 4 7 6	.	9 9 6 3		R	<CR>	<LF>
DST	:		2 8 5 3	.	0 0 1 2		R	<CR>	<LF>
	1	2	3	4	5	6	7	8	9

- 1 Colon
- 2 +/- sign, or Blank Space
- 3 2 to 7 places before the decimal point
- 4 Decimal point
- 5 1 to 6 places after the decimal point
- 6 Unit: blank space for mm, " for inches
- 7 **R** for Radius, **D** for Diameter display
- 8 Carriage return
- 9 Blank line (Line Feed)

Example 6: Probing function Circle Center

First center point coordinate, e.g. CCX = -1616.3429 mm. Second center point coordinate, e.g. CCY = +4362.9876 mm, (**C**ircle **C**enter **X** axis, **C**ircle **C**enter **Y** axis; coordinates depend on working plane).

Circle diameter DIA = 1250.0500 mm

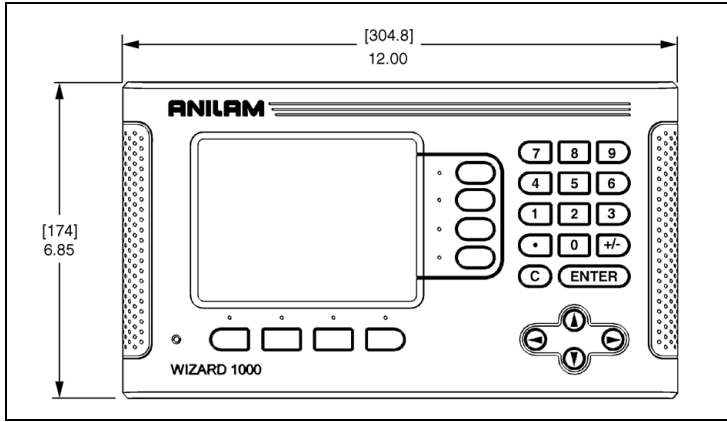
CCX	:	-	1 6 1 6	.	3 4 2 9		R	<CR>	<LF>
CCY	:	+	4 3 6 2	.	9 8 7 6		R	<CR>	<LF>
DIA	:		1 2 5 0	.	0 5 0 0		R	<CR>	<LF>
	1	2	3	4	5	6	7	8	9

- 1 Colon
- 2 +/- sign, or Blank Space
- 3 2 to 7 places before the decimal point
- 4 Decimal point
- 5 1 to 6 places after the decimal point
- 6 Unit: blank space for mm, " for inches
- 7 **R** for Radius, **D** for Diameter display
- 8 Carriage return
- 9 Blank line (Line Feed)

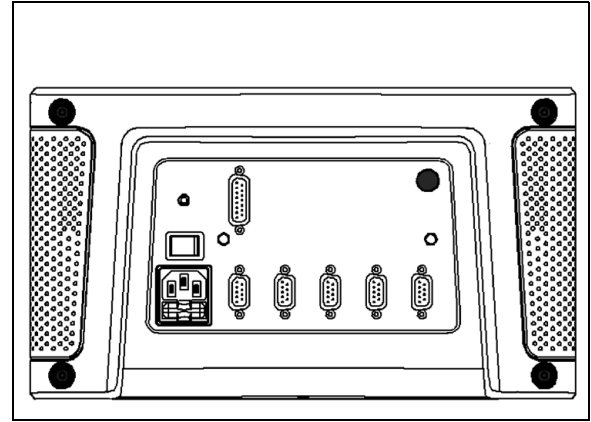
II - 6 Dimensions

DRO Dimensions

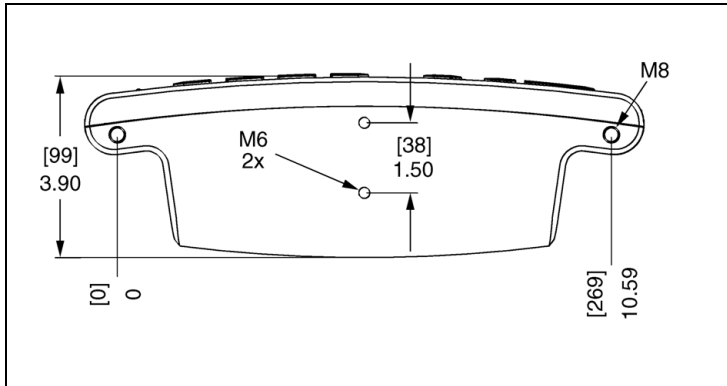
Dimensions in inches/mm



Front view with Dimensions



Back view



Bottom view with Dimensions

Numerics

1/2 soft key 20

A

Absolute 2

Absolute Distance Preset 17

Actual Value/Distance-To-Go soft key 2

Arc Milling 30

Automatic Non Linear error 58

Axis Labels 1

B

Backlash Compensation 59

C

Calling a Tool from the Tool Table (Turning) 36

Circle and Linear Pattern 20

Console adjustment 10

Counter Settings 60

D

Datum soft key (Milling) 14

Datum Soft Key (Turning) 36

Diagnostics 61

Diameter Axes (Milling) 5

Disable Ref soft key 3

Display Area 1

Display configuration 55

DRO Dimensions 71

E

Edge Finder (milling applications only) 5

Electrical Requirements 63

Enable Ref soft key 3

Enable/Disable Ref Function 3

Encoder Setup 53

Environmental Specifications 63

Error Compensation 55

G

Grounding (Earth) 63

I

Import/Export (setting) 10

Incline & Arc Milling 27

Incremental Distance Preset 19

Installation Setup Parameters 53

J

Job Clock 6

Job Setup Parameters 4

L

Language (setting) 10

Linear Error Compensation 56

M

Measured Value Output 6, 66

Mirror 5

N

Near Zero Warning 6

No Ref soft key 3

O

Operating Modes 2

P

Position Recall 61

Preset 17

Preventative Maintenance 63

Probing Functions 14

R

Radius/Diameter 39

Ref Symbols 1

Remote Switch 8

S

Scale factor 4

Screen Layout 1

Set Zero soft key 11

Setting Tool Offsets 34

Skew Compensation 7

Soft key Labels 1

Status Bar 1, 6

T

Taper Calculator 38

Tool Soft Key 12

Tool Table 12

Tool Table (Turning) 34

U

Units of measurement, setting 4

V

Vectoring 40
View Settings 8

Z

Z Coupling 40
Z/W Coupling 32

Acu-Rite Companies Inc.
is an
ISO 9001
CERTIFIED
MANUFACTURER



Acu-Rite Companies, Inc.
One Precision Way • Jamestown, NY 14701

