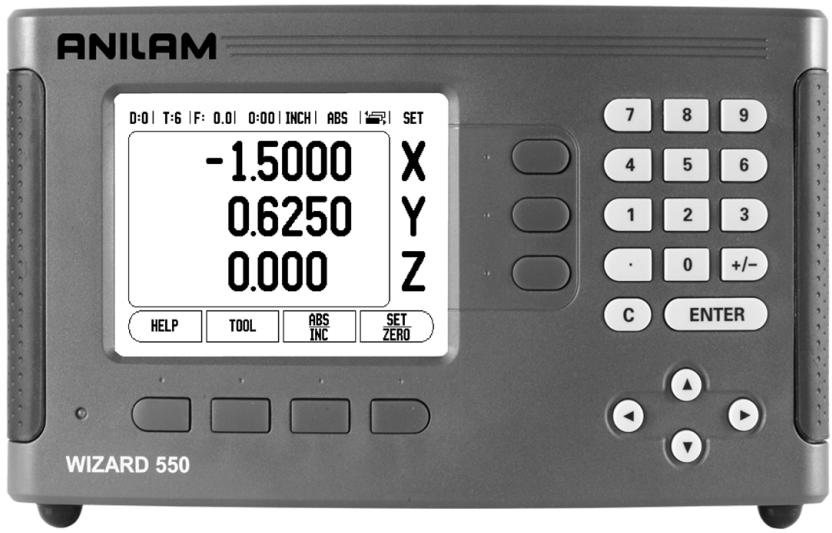
Wizard 550 READOUTS

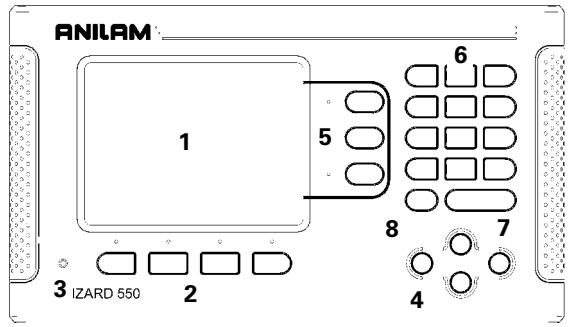


ANILAM

REFERENCE MANUAL

AMI
Automation and Metrology
8808 Tyler Blvd, Mentor, OH 44060
1-888-354-2525 | sales@auto-met.com
Request a quote: auto-met.com

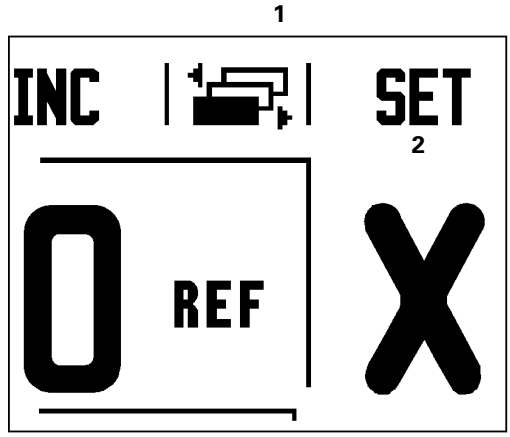
W550 Key Layout



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

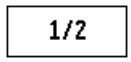
W550 Soft keys

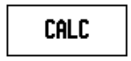
There are multiple pages of soft key functions to select from the operating modes. Use the LEFT/RIGHT arrow keys [**4**] to cursor through each page. The page indicator in the Status Bar will show the page orientation. The darkened page indicates the page you are currently on.



1 Page Indicator
2 Set Zero Indicator

Soft Key function (Page 1)	Soft key
Opens on-screen help instructions.	HELP
Opens the Tool Table. Page 21 for Milling, and page 46 for Turning.	TOOL
Switches display between operating modes Actual Value (Absolute) / Distance-to-Go (incremental). See page 12.	ABS INC
Toggles between Set/Zero functions. Used with individual axis keys (page 19).	SET ZERO

Soft Key function (Page 2)	Soft key
Opens the Datum form to set the datum for each axis (page 27).	
Opens the Preset form. This form is used to set a nominal position. This is a Distance-To-Go (Incremental) function (page 32).	
Used to divide the current position by two (page 35).	
Press to select the Circle Pattern, Linear Pattern, Incline Mill, or Arc Mill table (page 37).	

Soft Key function (Page 3)	Soft key
Opens the Job Setup menu and provides access to the Installation Setup soft key (page 12).	
Press when ready to identify a reference mark (page 11).	
Opens the Calculator functions for standard math, trigonometry, RPM, and Taper for Turning functions. The CALC key is also available on input forms where calculations may be required while inputting data.	
Toggles between inch and millimeter units (page 13).	

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.

Readout Parameter Access Code

Introduction

Software Version

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



This User's Manual covers the functions of the W550 for both **milling**, and **turning** applications. Operational information is arranged in three sections: General Operations, Mill Specific Operations, and Turn Specific Operations.

W550

DRO axis availability.



The W550 DRO is available in **two**, and **three axis** form. The 3 axis W550 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 W550.



Warning

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



Damage - Risk of electric shock

e.g. when opening a housing.

W550 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

Warranty

For information, go to www.anilam.com

Table of Contents

W550 Key Layout i
W550 Soft keys i

Readout Parameter Access Code

Access to Machine Parameter Operations iii

Introduction

Software Version v
W550 v
Symbols within Notes v
W550 Fonts v

Warranty

Warranty vii

I - 1 Fundamentals of Positioning

Datums 2
Actual Position, Nominal Position, and Distance-To-Go 2
Absolute Workpiece Positions 3
Incremental workpiece positions 3
Zero Angle Reference Axis 4
Reading head position 4
Encoder Reference Marks 5

I - 2 General Operations for W550

Screen Layout 6
General Navigation 7
General Overview 7
  Graphic Positioning Aid 7
Help Screen 8
Data Input Forms 9
  Instruction Box messages 9
  Error Messages 9
Power Up 10
Reference Mark Evaluation 10
  Working without reference mark evaluation 10
Enable/Disable Ref function 11
Operating Modes 12
Setup 12

Job Setup Parameters 13
Units 13
Scale Factor 13
Mirror 14
Edge Finder (milling applications only) 14
Diameter Axes 14
Measured Value Output 15
Near Zero Warning 15
Status Bar Settings 15
Job Clock 15
Skew Compensation (Milling applications only) 16
Remote Switch 17
Console Adjustment 18
Language 18
Import/Export 18
Set/Zero Soft Key Details 19
Calc Soft Key 19
RPM Calculator 20

I - 3 Milling Specific Operations

Soft Key Functions Detailed 21
Tool Soft Key 21
Tool Table 21
Import/Export 22
Tool Radius Compensation feature 23
Sign for the length difference ΔL 23
Entering tool data 24
Calling the Tool from the Tool Table 27
Datum Soft Key 27
Probing Functions for Datum Setting 27
Probing with a Tool 30
Example: Probe workpiece edge and set edge as datum 31
Presets 32
Absolute Distance Preset 32
Incremental Distance Preset 34
1/2 Soft Key 35

Features (Milling) 36
Circle, and Linear Patterns 37
Soft key Functions 37
Circle, and Linear Pattern Soft keys 38
Circle, or Linear Pattern Execution 39
Example: Enter data and execute a circle pattern 40
Incline, and Arc Milling 41
Soft Key Functions 41
Incline, and Arc Milling Soft keys 42
Incline Milling Form Entry 43
Arc Milling 44
Incline, and Arc Mill Execution 45

I - 4 Turning Specific Operations

Tool Display Icon 46
Tool Table 46
Datum Setting 48
Taper Calculator Soft Key 50
Presets 51
Radius/Diameter Soft Key 51
Vectoring 52
Z Coupling 52
Disabling Z0, and Z Coupling 53

II - 1 Installation Setup

Installation Setup Parameters 56
Encoder Setup 57
Display Configuration 59
Error Compensation 59
Linear Error Compensation 60
Non-Linear Error Compensation 60
Setup procedure for Non-linear Error 61
Starting a Non-Linear Error Compensation Table 62
Configuring the Compensation Table 63
Automatic Non Linear error compensation 63
Backlash Compensation 65
Counter Settings 65
Diagnostics 66
Keypad Test 66
Edge Finder Test 66
Display Test 66

II - 2 Data Interface

Serial Port 68

II - 3 Installation and Electrical Connections

Installation 69
Electrical requirements 69
Environmental 69
Preventative maintenance 69

II - 4 I/O Connections

Wiring the Serial communication cable 71
Signal 71

II - 5 Remote Switch Data Output

Data output using external signals 72
Data output using Edge Finder 74

II - 6 Error Messages

..... 77

II - 7 Dimensions

DRO Dimensions 79

Operating Instructions

W550

I - 1 Fundamentals of Positioning

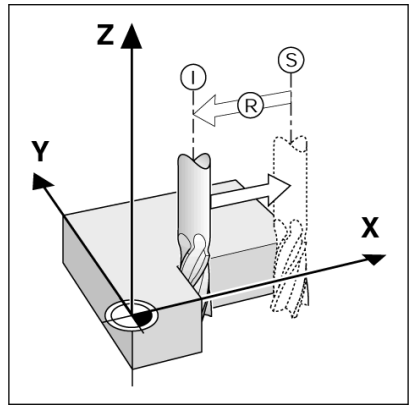
Datums

The workpiece drawing identifies a certain point on the workpiece (example: "a corner") as the absolute datum, and perhaps one, or more other points as relative datums.

The datum setting procedure establishes these points as the origin of the absolute, or relative coordinate systems. The workpiece, which is aligned with the machine axes, is moved to a certain position relative to the tool. The display is set either to zero, or to another appropriate value (e.g., to compensate for tool radius).

Actual Position, Nominal Position, and Distance-To-Go

The position of the tool at any given moment is called the ACTUAL POSITION **I**, while the position that the tool is to move to is called the NOMINAL POSITION **S**. The distance from the nominal position to the actual position is called the DISTANCE-TO-GO **R**.

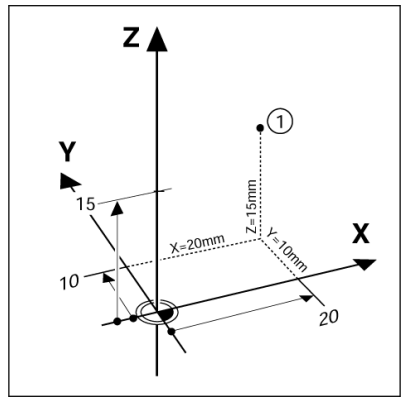


Absolute Workpiece Positions

Each position on the workpiece is uniquely identified by its absolute coordinates.

Example: Absolute coordinates of position **1**:
X = 20 mm
Y = 10 mm
Z = 15 mm

If you are drilling, or milling a workpiece according to a workpiece drawing with **absolute coordinates**, the tool is moving to the value of the coordinates.



Incremental workpiece positions

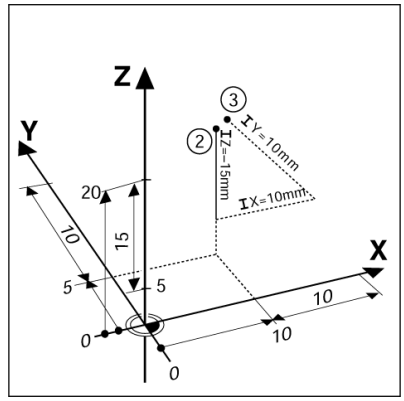
A position can also be referenced to the preceding nominal position. In this case the relative datum is always the last nominal position. Such coordinates are referred to as **incremental coordinates** (increment = increase). They are also called incremental, or chain dimensions, since the positions are defined as a chain of dimensions. Incremental coordinates are designated with the prefix **I**.

Example: Incremental coordinates of position **3** referenced to position **2**.

Absolute coordinates of position **2**:
X = 10 mm
Y = 5 mm
Z = 20 mm

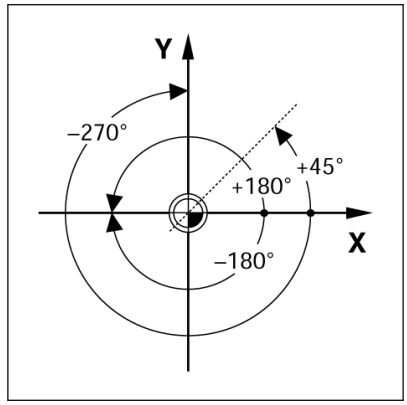
Incremental coordinates of position **3**:
IX = 10 mm
IY = 5 mm
IZ = 20 mm

If you are drilling, or milling a workpiece according to a drawing with incremental coordinates, you are moving the tool **by** the value of the coordinates.



Zero Angle Reference Axis

The Zero Angle Reference Axis is the 0.0° position. It is defined as one of the two axes in the plane of rotation. The following table defines the Zero Angle where the position of the angle is zero for the three possible planes of rotation.



For angular positions, the following reference axes are defined:

Plane	Zero Angle Reference Axis
XY	+X
YZ	+Y
ZX	+Z

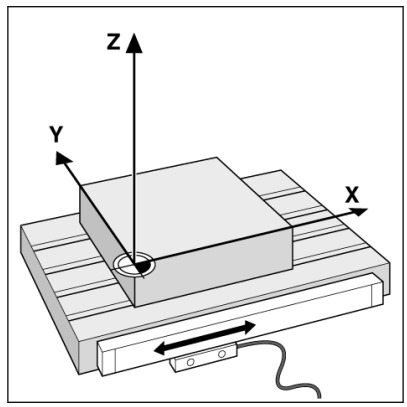
Positive direction of rotation is counterclockwise if the working plane is viewed in the negative tool axis direction.

EXAMPLE: Angle in the working plane X / Y

Plane	Zero Angle Reference Axis
+ 45°	... bisecting line between +X and +Y
+/- 180°	... negative X axis
- 270°	... positive Y axis

Reading head position

The reading head position provides feed back to the W550 that converts the movement of the machine axes into electrical signals. The W550 constantly evaluates these signals, and calculates the actual positions of the machine axes, which it displays as a numerical value on the screen.



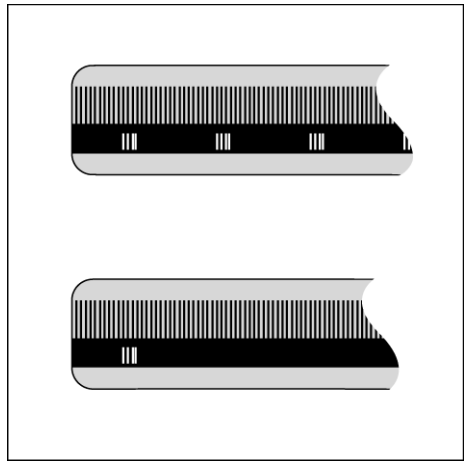
If there is an interruption in power, the calculated position will no longer correspond to the actual position. When power is restored, you can re-establish this relationship by using the reference marks on the provided on the scale. The W550 provides the Reference Mark Evaluation feature (**REF**).

Encoder Reference Marks

Encoders normally contain one or more reference marks which the W550 Reference Mark Evaluation feature uses to re-establish datum positions after a power interruption. There are two main options available for reference marks; fixed and distance-coded.

Position Trac (Distance-coded reference marks)**:** Encoders that have marks separated by a specific encryption pattern allows the W550 to use any two pair of marks along the length of the encoder to re-establish the prior datums. This configuration means that the operator only has to travel less than two inches any where along the encoder, to re-establish the datums when the W550 is turned back on.

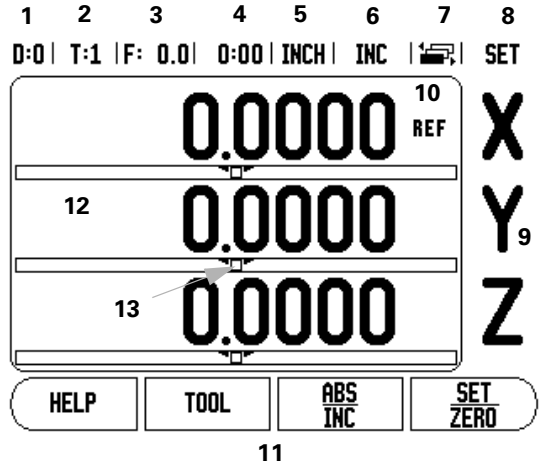
Fixed reference marks: Encoders that have one or more marks on fixed intervals, have to re-establish the datums correctly. It is necessary to use the same exact reference mark, during the Reference Mark Evaluation routine, that was used when the datum was first established.



The established datums' cannot be restored from one power cycle to the next if the reference marks were not crossed before the datums were set

I - 2 General Operations for W550

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)

The Anilam W550 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.

General Navigation

- Use key pad to enter numeric values within each field.
- The ENTER key will confirm the entry within a field, and return to the previous screen.
- Press the C key to clear entries, and error messages, or return back to the previous screen.
- SOFT KEY labels show the various milling, or turning functions. These functions are selected by pressing the corresponding soft key directly below each soft key label. There are 3 pages of selectable soft key functions. These are accessed using the LEFT or RIGHT arrow keys as indicated below.
- LEFT or RIGHT arrow keys move through pages 1-3 of soft key selectable functions. The current page will be highlighted in the Status bar at the top of screen.
- Use the UP, or DOWN arrow keys to move between fields within a form, and list boxes within a menu. The orientation of the cursor is such that it will return to the top once it has reached the bottom of a menu.

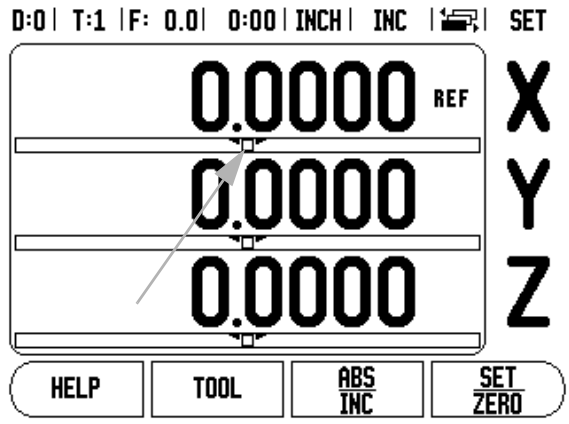
General Overview

Graphic Positioning Aid

When traversing to display value zero (in the Distance-To-Go mode), W550 displays a graphic positioning aid.

W550 displays the graphic positioning aid in a narrow rectangle underneath the currently active axis. Two triangular marks in the center of the rectangle symbolize the nominal position.

A small square symbolizes the axis slide. An arrow indicating the direction appears in the square while the axis is moving towards, or away from the nominal position. Note that the square does not begin to move until the axis slide is near the nominal position. Setting up the graphic positioning aid, see "Status Bar Settings" on page 15 under Job Setup.



Help Screen

The integrated operating instructions provide information and assistance in any situation.

To call the operating instructions:

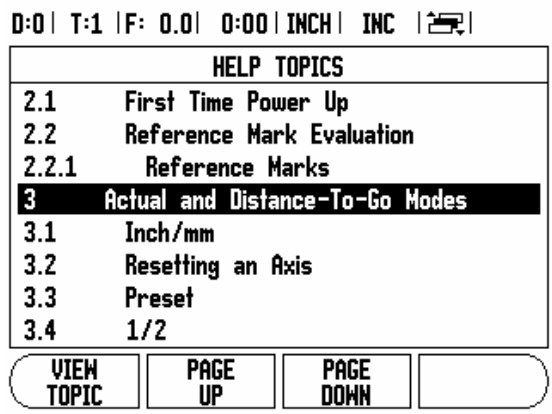
- Press the HELP soft key.
- Information relevant to the current operation will be displayed.
- Use the UP/DOWN arrow keys if the explanation is spread over more than one screen page.

To view information on another topic:

- Press the LIST OF TOPICS soft key.
- Press the UP/DOWN arrow keys to scroll through the index.
- Press the ENTER key to select the item you need.

To exit the operating instructions:

- Press the C key.



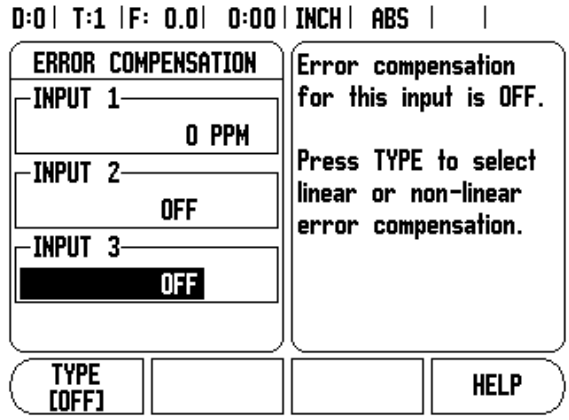
Data Input Forms

Information required for various operational functions and setup parameters are entered through a data input form. These forms will appear after selecting features that require any additional information. Each form provides specific fields for entering the required information.

Changes must be confirmed by pressing the ENTER key for them to become effective. If you do not want to save your changes, press the C key to return to the previous screen without saving changes.
In some cases such as the Tool Table, the C key is used in place of the ENTER key.

Instruction Box messages

Whenever a Menu, or Form is opened, an instruction box will also open immediately to the right of it. This message box will provide information for what the chosen function does, and present instructions for available options.



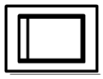
Error Messages

If an error occurs while working with the W550, the message will appear on the display, and provide an explanation of what caused the error. See "Error Messages" on page 77.

To clear the error message:

- ▶ Press the C key.
- ■ Error message will be cleared, and normal operation can be continued.

Power Up

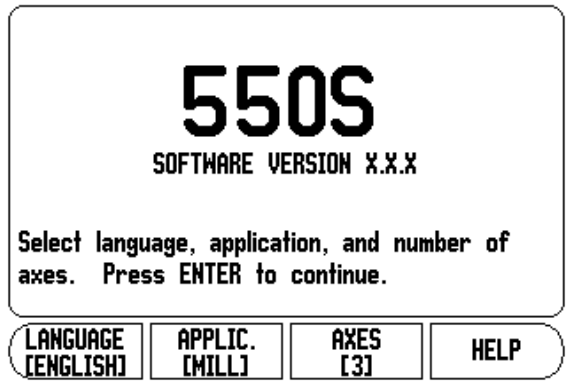


Switch on the power (located on the back). The initial screen will appear. This screen will only appear the very first time the unit is powered up. The following steps may have already been completed by the installer.

- Select the proper language by pressing the LANGUAGE soft key.
- Choose your application of either MILL or TURN. The APPLIC. [MILL/TURN] soft key toggles between these two settings.
- Next, select the number of axes required. When complete, press the ENTER hard key.

If necessary, you can change the application later in Installation Setup under Counter Settings.

First-Time Configuration Screen



The W550 is now ready for the remaining setup requirements. It is now in the operating mode "Absolute". Each active axis will have a flashing"REF"sign next to it. The following section, "Reference Mark Evaluation", describes setting up this feature.

Reference Mark Evaluation

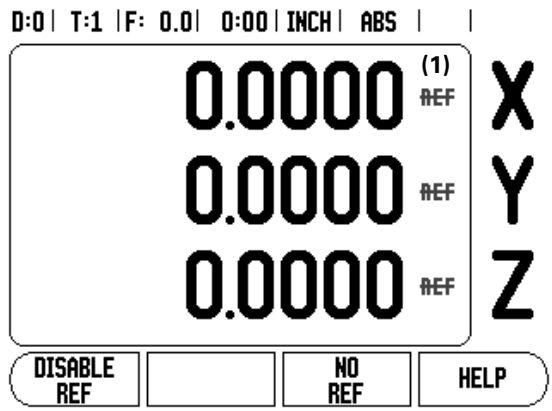
The W550 reference mark evaluation feature (**1**), automatically re-establishes the relationship between axis slide positions and display values that was last defined by setting the datum.

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 W550 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 W550 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.

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 W550 will return to the DRO display screen automatically.

Operating Modes

The W550 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.

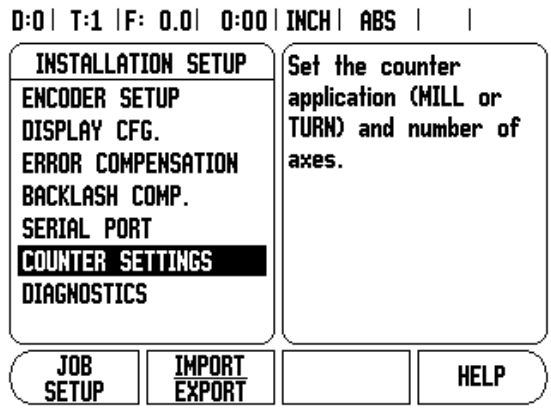
The Turning application provides a quick method for coupling the Z axes positions on a 3 axis system. For more information, see "Z Coupling" on page 52.

Setup

W550 offers two categories for setting up operating parameters. These categories are: Installation Setup and Job Setup. The Job Setup parameters are used to accommodate specific machining requirements for each job. Installation Setup is used to establish encoder, display and communication parameters.

The Installation Setup menu is accessed by pressing the SETUP soft key, then the INSTALLATION SETUP soft key. When in the Installation Setup menu, the following soft keys will be available:

- **JOB SETUP**: Press to begin accessing the Job Setup parameters.
- **IMPORT/EXPORT**: Press to begin importing or exporting operating parameters. See "Import/Export" on page 18.
- **HELP**: Will open on-line help.



- To view, and change Installation Setup parameters, first press the SETUP soft key, then the INSTALLATION SETUP soft key.
- Use the UP/DOWN arrow keys to highlight the parameters of interest.
- press the ENTER key.

Job Setup Parameters

- To view and change Job Setup parameters, first press the SETUP soft key.
- Use the UP/DOWN arrow keys to highlight the parameters of interest.
- press the ENTER key.

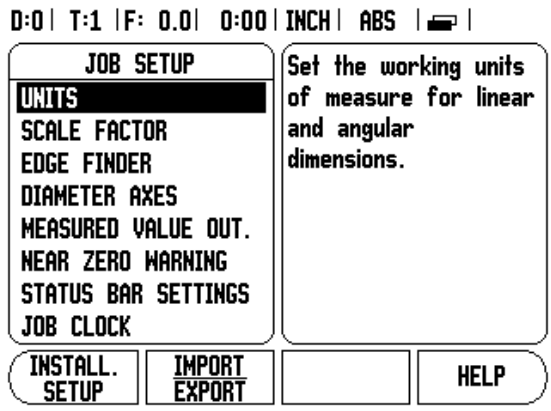
The Job Setup Data can be imported, or exported using the IMPORT/EXPORT soft key

Exporting the current Job Setup:

- Press the SETUP soft key.
- Press the IMPORT/EXPORT soft key.
- Press the EXPORT soft key.

Importing a new Job Setup Table

- Press the SETUP soft key.
- Press the IMPORT/EXPORT soft key.
- Press the IMPORT soft 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 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)

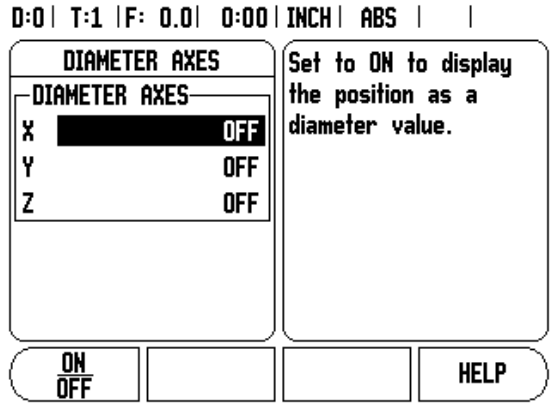
The diameter, length offset and units of the edge finder are set in this form. Both values are in the units indicated in the form. Please see "Probing Functions for Datum Setting" on page 27 for details on using Edge Finder functions.

- 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 51 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.



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 W550 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 72" 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 27. 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.

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 W550 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 using the serial port. See "Data Interface" on page 67.

- 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

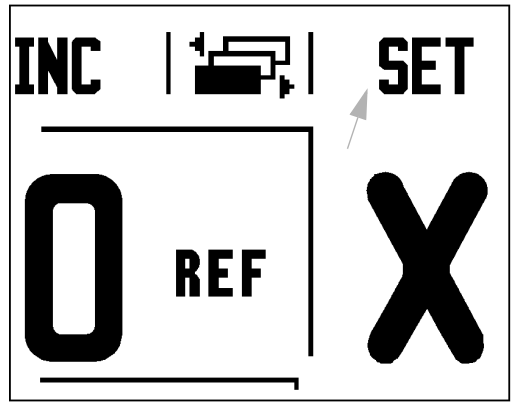
The SET/ZERO soft key is a key that determines the effect of pressing an Axis key. This key is a toggle key, switching the functionality between Set, and Zero. The current state is indicated in the Status Bar shown as "SET" in this view.

When the state is Set, and the W550 is in Actual Value mode, selecting an Axis key opens the Datum form for the selected axis. If the W550 is in Distance-To-Go mode, a Preset form opens.

When the state is Zero, and the W550 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 W550 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.



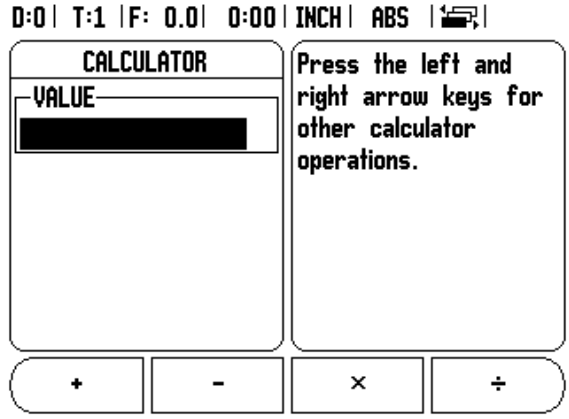
Calc Soft Key

W550's calculator is capable of handling everything from simple arithmetic to complex trigonometry and RPM calculations.

Press the CALC soft key to access the STANDARD/TRIG and RPM soft keys. The CALC soft key is also available on input forms where preforming calculations may be required while inputting data.



When you need to enter more than one calculation into a numeric field, the calculator will perform multiplication and division before it performs addition and subtraction. If you were to enter 3 + 1 ÷ 8, W550 will divide one by eight, then add three for an answer of 3.125.



Trig functions contain all trig operators as well as, square and square root. Whenever you're calculating the SIN, COS or TAN of an angle, enter the angle first and then press the appropriate soft key.



Angle values use the current angle format selection of decimal degrees, DMS, or radians.

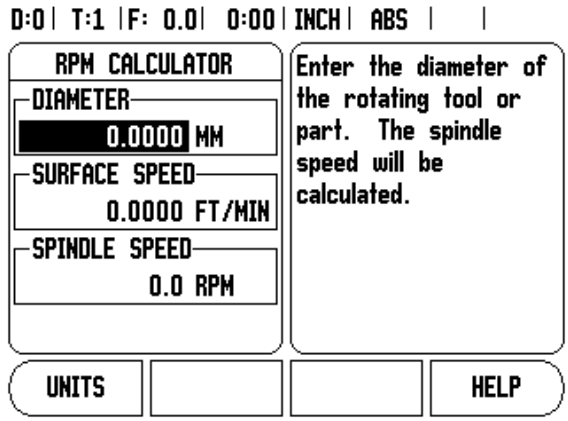
RPM Calculator

The RPM calculator is used to determine the RPM (or surface cutting speed) based on a specified tool (part, for turning applications) diameter. The values shown in this Figure are only an example. Consult your tool manufacturer's manual to verify spindle speed ranges per tool.

- Press CALC.
- Press the RPM soft key to open the RPM Calculator form.
- The RPM calculator requires a tool diameter. Use the numeric hard keys to enter a diameter value. The diameter value will default to the current tool's diameter. If there is no last value entered in this power cycle, the default value is 0.
- If a surface speed value is required, enter the value using the numeric hard keys. When a surface speed value is entered, the associated RPM value will be calculated.

When in the Surface Speed field, a soft key is available for opening on-line help. The table may be consulted for a recommended range of surface speeds for the material being machined.

- Press the UNITS soft key to show the units as inch or millimeter.
- The RPM Calculator form is closed by pressing the C key saving current data.



I - 3 Milling Specific Operations

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

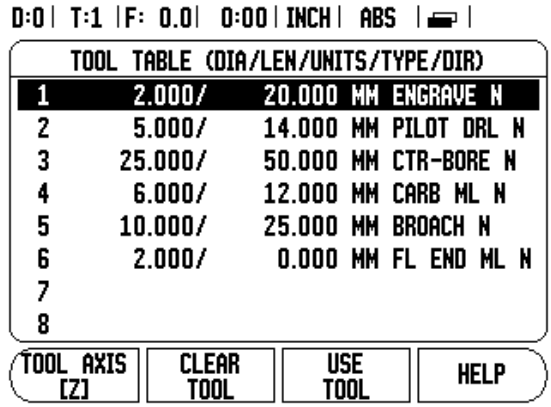
Soft Key Functions Detailed

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 W550 can store up to 16 tools within the tool table.

Tool Table

The W550 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.



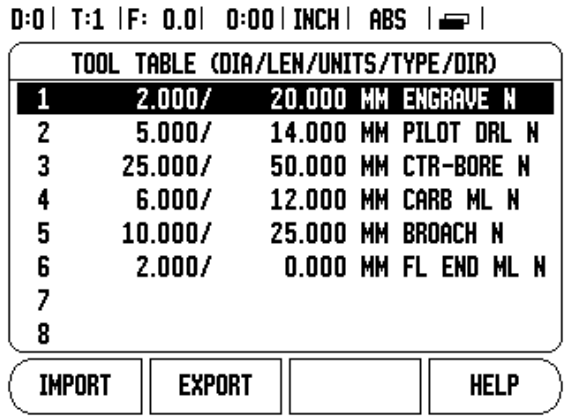
The following soft keys are also available while in the Tool Table form, or in the individual tool data form:

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.	TOOL AXIS [Z]
Press to automatically enter the tool offset length. This key is only available in the TOOL LENGTH field.	TEACH LENGTH
The Tool Types form is opened using this key, and is only available in the Type field.	TOOL TYPES
To remove a tool from the table, use the CLEAR TOOL key.	CLEAR TOOL
To use a tool from the table, highlight the tool, and press the USE TOOL key.	USE TOOL
Press access available help files.	HELP

Import/Export

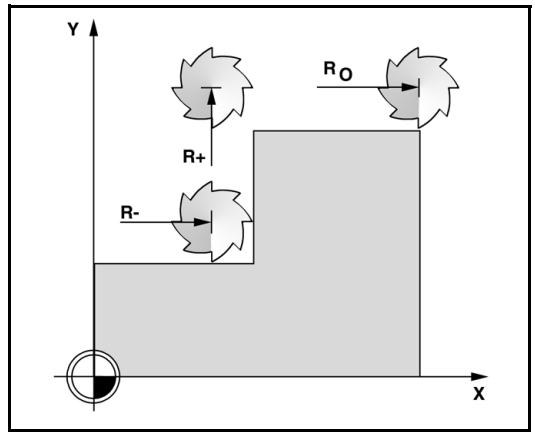
Tool Table information can be imported, or exported over the serial port.

- IMPORT, and EXPORT soft keys are available in the Tool Table screen.
- Press IMPORT to download a Tool Table from a PC.
- Press EXPORT to upload the Tool Table to a PC.
- To exit, press the C key.



Tool Radius Compensation feature

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 32.

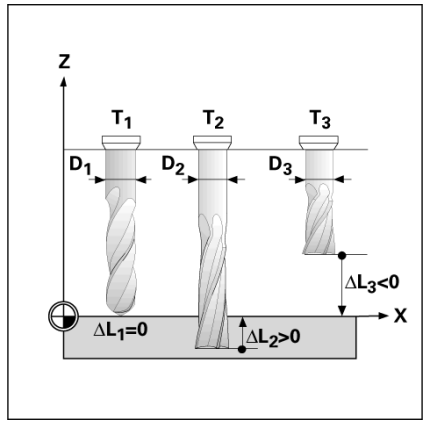


The length offset may be entered as a known value or the W550 may determine the offset automatically. The tool length is the difference in length ΔL between the tool and the reference tool. The length difference is indicated with the "Δ"symbol. The reference tool is indicated by T1.

Sign for the length difference ΔL

If the tool is **longer** than the reference tool: ΔL > 0 (+).

If the tool is **shorter** than the reference tool:ΔL < 0 (–).



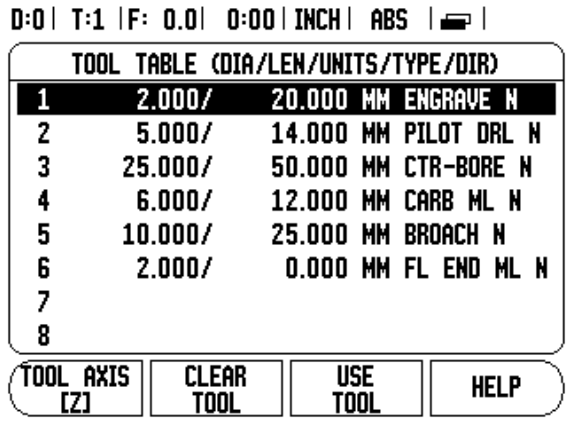
The length offset may be entered as a known value, or the W550 may determine the offset automatically. To have the W550 determine a tool's length offset involves touching the tip of each tool to a common reference surface. This allows the W550 to determine the difference between the length of each tool.

Move the tool until it's tip is touching the reference surface. Press the TEACH LENGTH soft key. The W550 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.

Entering tool data

- Choose TOOL soft key.
- Cursor to the desired tool and press ENTER. The Tool Description form will appear.
- Enter the tool diameter.
- Enter the tool length or press TEACH LENGTH.
- Enter the tool units (inch/mm).
- Enter the tool type.
- Spindle data is not required unless the Spindle Speed Control has been installed. If installed, then refer to the CSS I/O User's Manual.
- Press C to return to the tool table. Press C to exit.



Tool Table Usage

Example: Entering the tool length and diameter into the tool table.

Tool diameter 2.00

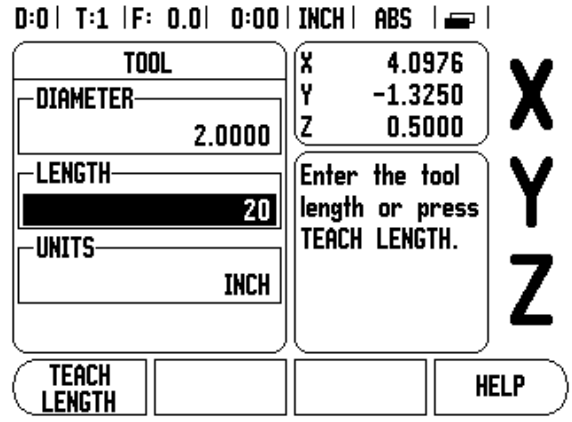
Tool length: 20.000

Tool unit: mm

Tool type: flat end mill

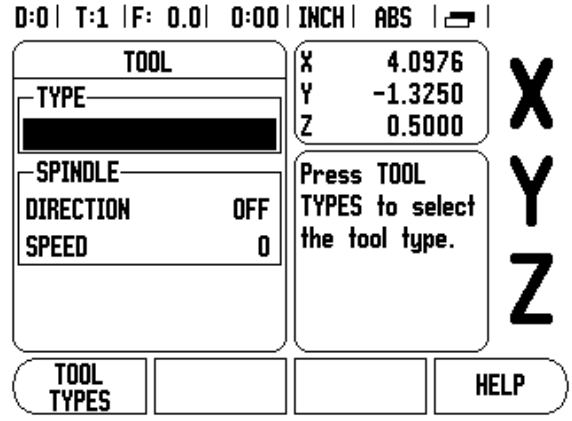


It is also possible to have W550 determine the length of an offset. See - ALTERNATIVE METHOD -.





Spindle Speed Control information is only required when CSS I/O box is installed. Refer to the CSS I/O User Manual if this has been installed.



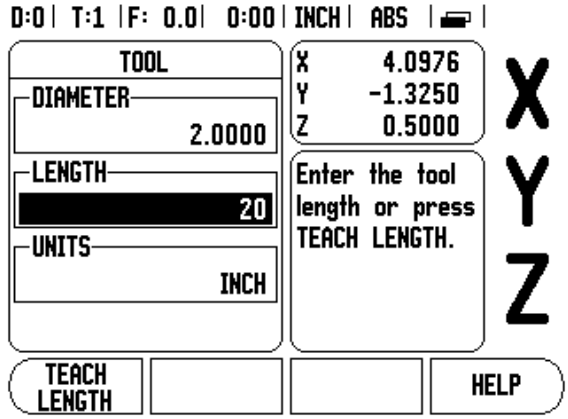
Entering a tool:

- Press the TOOL. soft key. The cursor will default to the Tool Table form.
- Cursor to the tool you want to define or enter the tool number. Press ENTER.
- Enter the tool diameter, for example (2).
- Press the DOWN ARROW key.
- Enter the tool length, for example, (20).
- Press the DOWN ARROW key.

-ALTERNATIVE METHOD -

It is also possible to have W550 determine an offset. This method involves touching the tip of each tool to a common reference surface. This allows W550 to determine the difference between the length of each tool.

- Move the tool until its tip is touching the reference surface.
- Press the TEACH LENGTH soft key. W550 will calculate an offset relative to this surface.
- Repeat the procedure for each additional tool using the same reference surface.
- Press ENTER.



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

If the tool table already contains tools in which the length has been set, the reference surface should first be established using one of them. If not, you will not be able to switch between the new tools and the existing tools without having to re-establish the datum. Before adding the new tools, select one of the tools from the tool table. Touch the tool to a reference surface and set the datum to 0

TOOL UNIT

- Enter the tool unit (inch/mm).
- Cursor to the Tool Type field.

TOOL TYPE

- Press TOOL TYPES soft key.
- Press ENTER.

Calling the Tool from the Tool Table

Before you start machining, select the tool you are using from the tool table. W550 then takes into account the stored tool data when working with tool compensation.

Tool call

- 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 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 W550 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 W550 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, W550 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 13..

Example: Set centerline between two workpiece edges as datum

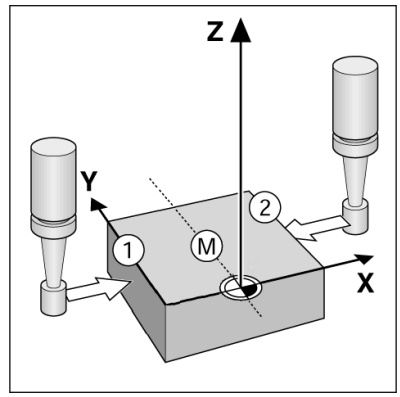
0 is offered as a default value for the coordinate. Enter the desired coordinate for the workpiece edge. For this example Y = 0.0, set the coordinate as a datum for this workpiece.

▶ Press the ENTER hard key.

The position of the centerline **M** is determined by probing the edges **1**, and **2**.

The centerline is parallel to the Y axis.

Desired coordinate of the centerline: X = 0.0



Spacing between edges is displayed on the message box when using the probe centerline feature.

- Press the DATUM soft key.
- Press the DOWN arrow key to select the item you need.
- Press the PROBE soft key.

Probe the 1st edge **X**

- Move the edge finder toward workpiece edge **1** until the LEDs in the edge finder light up.

Probe the 2nd edge in **X**

- Move the edge finder toward workpiece edge **2** until the LEDs in the edge finder light up. The distance between the edges is displayed on the message box.
- Retract the edge finder from the workpiece.

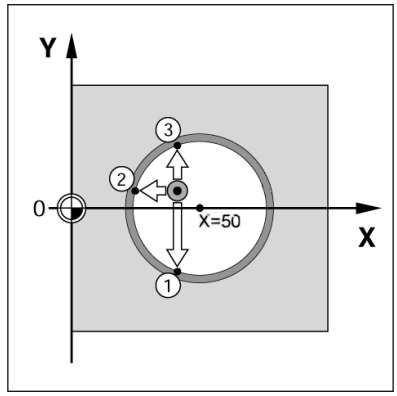
Enter the value for **X**

- Enter coordinate (X = 0.0), and transfer coordinate as datum for the centerline.
- Press the ENTER hard key.

Example: Probe the center of a hole with an edge finder, and set the datum 2.00 from the circle's center

X coordinate of the circle center: X = 2.00

Y coordinate of the circle center: Y = 2.00



- Press the DATUM soft key.
- Press the DOWN arrow key to select the item you need.
- Press the PROBE soft key.
- Press the CIRCLE CENTER soft key.
- Move the edge finder toward the first point **1** on the circumference until the LEDs in the edge finder light up.
- Probe two additional points on the circumference in the same manner. Further instructions appear on the screen. See Instruction Box for the measured diameter.

Enter center point **X** X = 2.00

- Enter the first coordinate (X = 2.00).
- Press the DOWN arrow hard key.

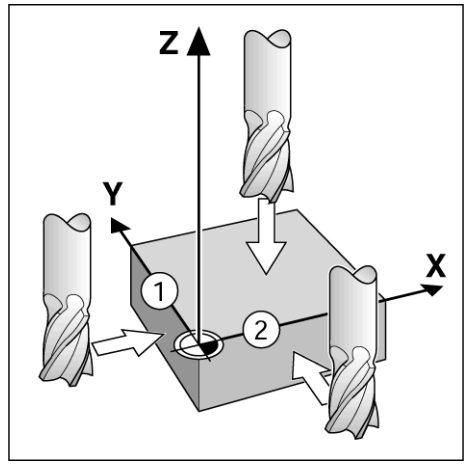
Enter center point **Y** Y = 0.00

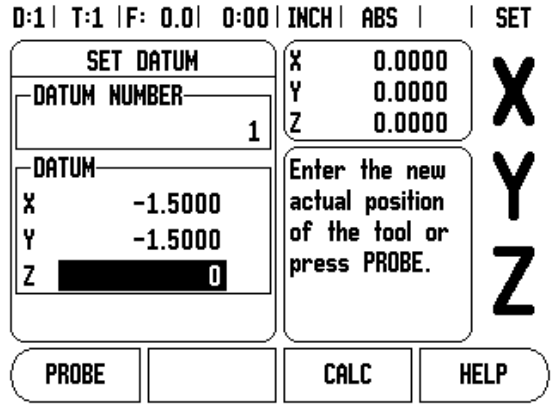
- Accept default entry Y = 2.00.
- Press the ENTER hard key.

Probing with a Tool

If using a tool, or non-electrical edge finder to set datum points, the W550 probing functions can still be used.

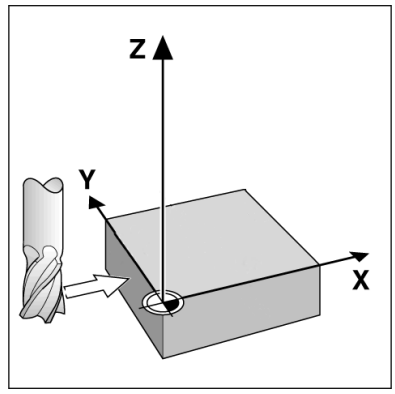
Datum points can be set by touching the edges of a workpiece, one after the other with a tool, and then manually entering the tool's position as datum points.

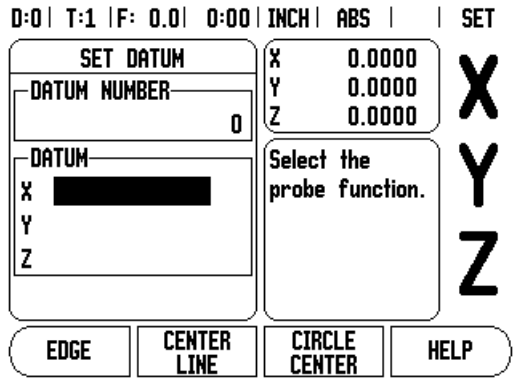




Example: Probe workpiece edge and set edge as datum

- 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.

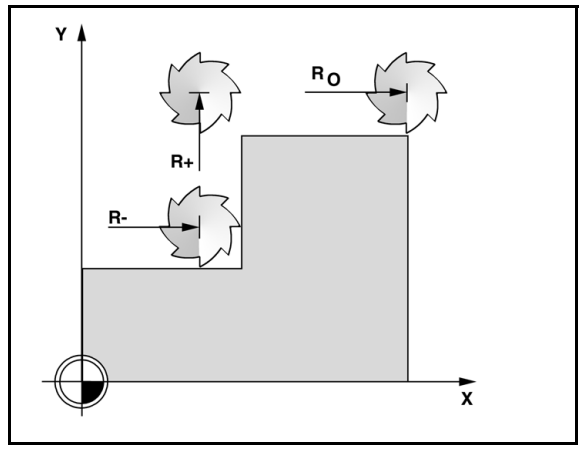




Presets

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.

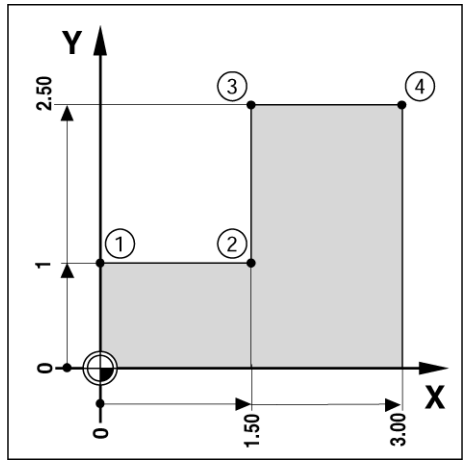


Absolute Distance Preset

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:

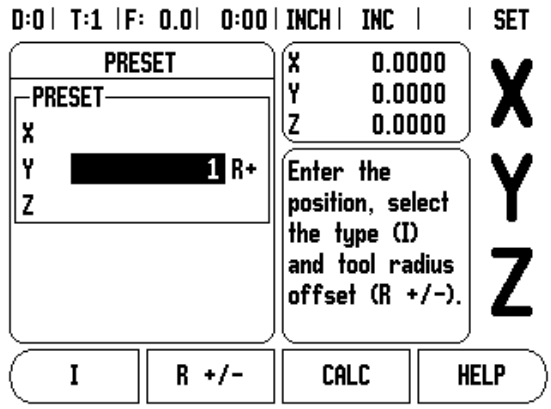
- Select the tool with the appropriate tool data.
- Pre-position the tool to an appropriate location (such as X = Y = -1″).
- Move the tool to milling depth.
- Press the PRESET soft key.
- Press the Y axis key

-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.



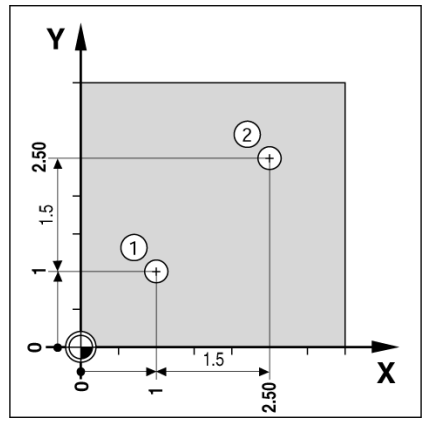
Incremental Distance Preset

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

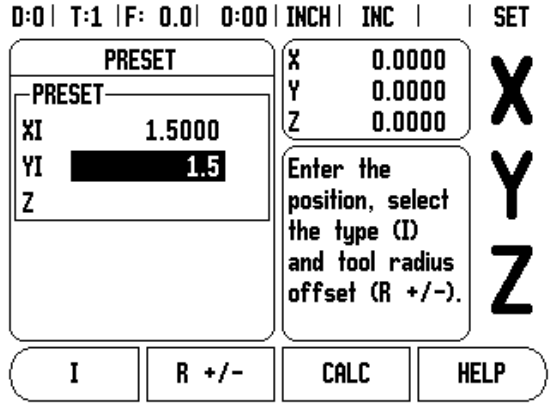


Enter the coordinates in incremental dimensions. These are indicated in the following (and on the screen) with a preceding **I** (Incremental). The datum is the workpiece zero.

- 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.



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.

Features (Milling)

Pressing the FEATURES soft key will provide access to the Circle Pattern, Linear Pattern, Incline Mill, and Arc Mill milling features.

The W550 provides one user definable pattern each of these features. They can be recalled, and executed from the DRO anytime during operation.

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

Function	Soft key
Press to access the circle pattern table.	
Press to access the linear pattern table.	
Press to access the Incline Mill entry form.	
Press to access the Arc Mill entry form.	

The Circle Pattern, and Linear Pattern features provide ways to calculate, and machine various hole patterns. 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.



Defined Circle, Linear, Incline, and Arc patterns are remembered when power is cycled

Circle, and Linear Patterns

This section describes the Circle, and Linear pattern tables, and capabilities. The W550 provides storage of 1 user definable pattern 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.

The View Graphic enables verification of the hole pattern before before any machining is started. 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 HELP for additional information about the pattern.	

Circle, and Linear Pattern Soft keys

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

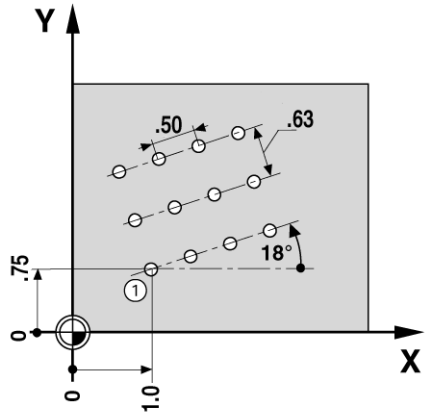
Function	Soft key
Press HELP for additional information about the pattern.	HELP
Press to use the current absolute position.	TEACH
Press to open the Calculator for standard math, and trigonometry functions.	CALC

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

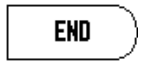
- Type: type of pattern, ARRAY or FRAME.
- First Hole: location of the first hole of the pattern.
- Holes per Row: number of holes in each row of a pattern.
- Angle: the angle, or rotation of the pattern.
- Depth: the target depth for drilling in the tool axis.
- Number of Rows: the number of rows in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



Circle, or Linear Pattern Execution

- To execute a circle, or linear pattern, enter the data in the pattern form to be executed.
- Press the ENTER key.
  - The W550 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.	

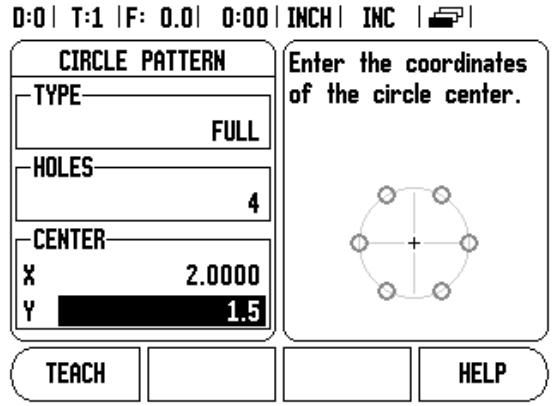


Press the VIEW soft key to toggle the views between distance-to-go positions, graphic view, and absolute positions.

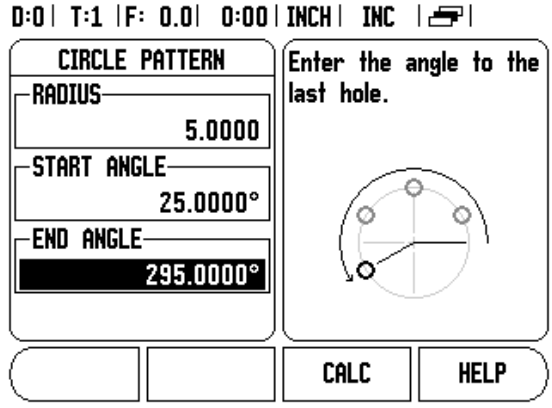
Example: Enter data and execute a circle pattern.

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
- Execute the feature by pressing the NEXT HOLE / PREVIOUS HOLE soft key. To cancel, or end the feature press the END soft key.



Incline, and Arc Milling

This section describes the Incline Mill, and Arc Mill tables, and capabilities. The W550 provides storage of 1 user definable pattern each for Incline Mill, and Arc Mill. Once patterns are defined, they are remembered when power is cycled. They can be recalled, and executed from the DRO until removed.

The View Graphic enables verification of the pattern before any machining is started.

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

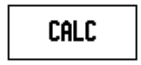
Soft Key Functions

The following soft keys are available while in the Incline or Arc Mill forms.

Function	Soft key
Press to access the FEATURES soft keys.	FEATURES
Press to access the Incline Mill Table.	INCLINE MILL
Press to access the Arc Mill Table.	ARC MILL
Press HELP for additional information about the pattern.	HELP

Incline, and Arc Milling Soft keys

The following additional soft keys are available while in the Incline, and Arc Milling feature.

Function	Soft key
Press HELP for additional information about the pattern.	
Press to use the current absolute position.	
Press to open the Calculator for standard math, and trigonometry functions.	

Incline Milling Form Entry

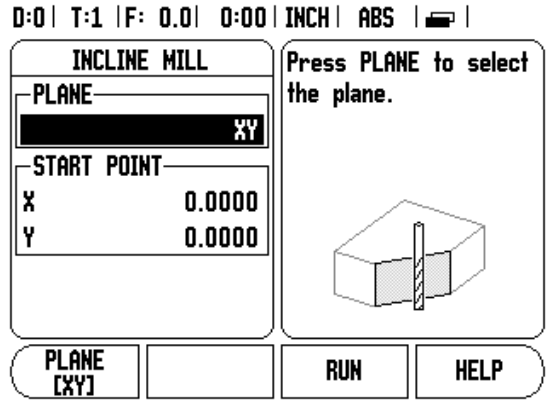
The Incline Milling form is used to specify the flat surface to be milled. The data is entered in the "Incline Mill Form". Select INCLINE MILL soft key to define the feature. The following listed fields require data input.

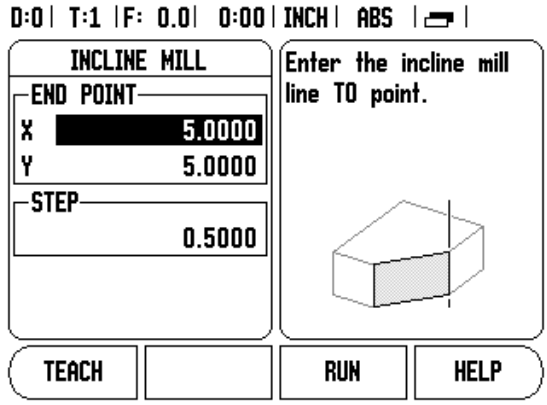
- **Plane:** Select the plane by pressing the PLANE soft key. The current selection is shown on the soft key and in the plane field. The graphic in the message box aids in selecting the correct plane.
- **Start Point:** Enter the coordinates of the start point, or press TEACH to set the coordinate to 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 execute the feature. Press C hard key to exit the form without saving the feature.

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





Arc Milling

The Arc Milling form is used to specify the curved surface to be milled. The data is entered in the "Arc Mill Form". Select ARC MILL soft key to define the feature.

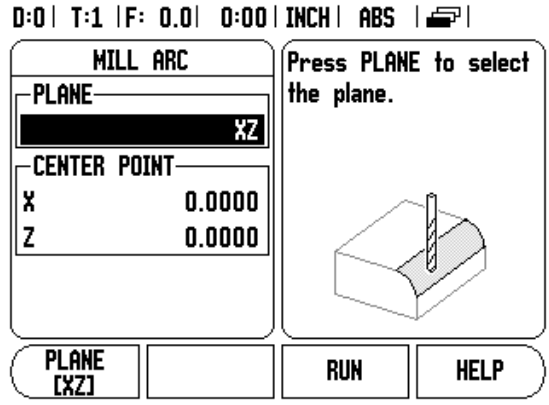
The following listed fields require data input.

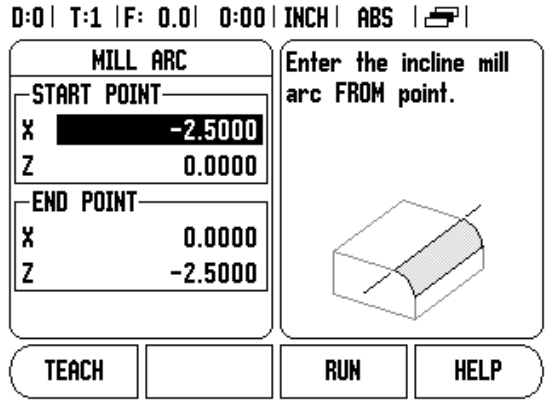
- **Plane Selection:** Select the plane by pressing the PLANE soft key. The current selection is shown on the soft key, and in the plane field. The graphic in the message box aids in selecting the correct plane.
- **Center Point:** Enter the coordinates of the arc's center point, or press TEACH to set 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.

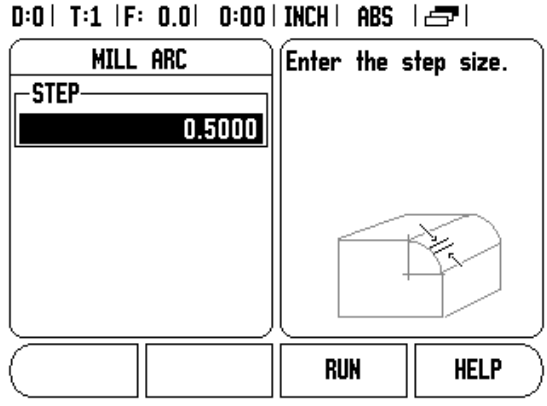
The Step size is optional. If the value is zero, the operator decides at run-time how far to move between each step.

Press the ENTER key to close the form and execute the feature. Press C hard key to exit the form without saving the feature.

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







Incline, and Arc Mill Execution

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 cross hair representing the tool is on the line representing the surface, the tool is in position. The tool cross hair 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.

I - 4 Turning Specific Operations

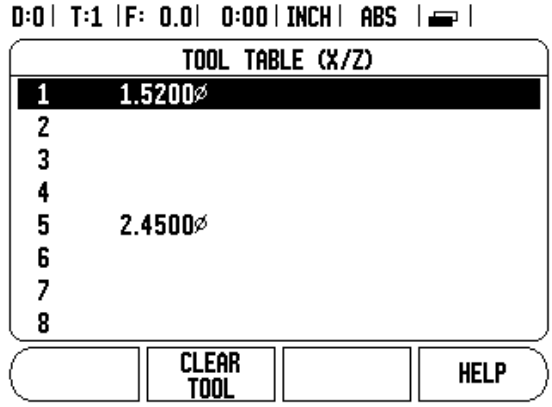
This section discusses operations specific to turning applications only.

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 W550 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.

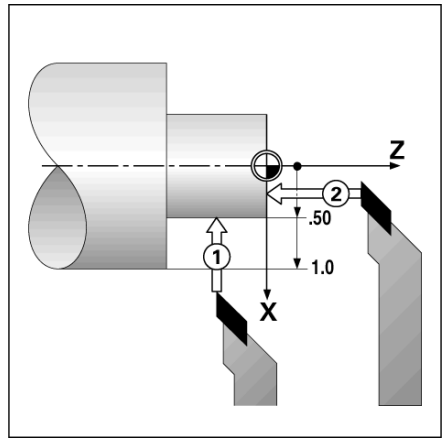


Setting Tool Offsets Example 1: Using TOOL/SET

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.



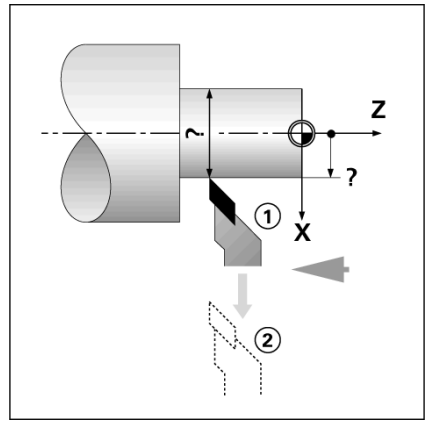
- Press the ENTER key.
- Select the axis (X) key.
- Enter the position of the tool tip, for example, X=.100. Remember to ensure the W550 is in diameter display mode (Ø) if the input is a diameter value.
- Touch the workpiece face with the tool.
- Cursor to the Z axis (2), then set the position display 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.

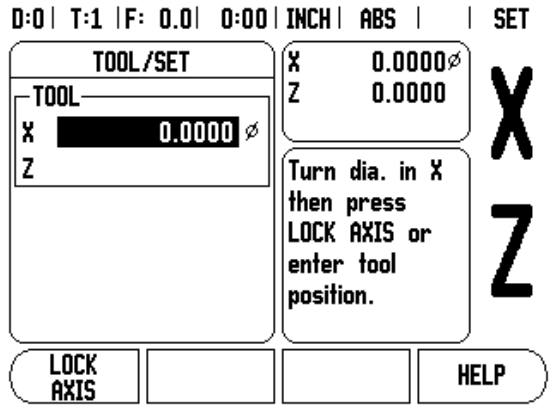
To use the LOCK AXIS function:



- 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 W550 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.



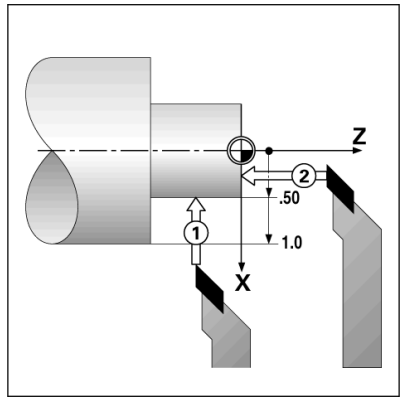
Datum Setting

See "Datum Soft Key" on page 27. 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.

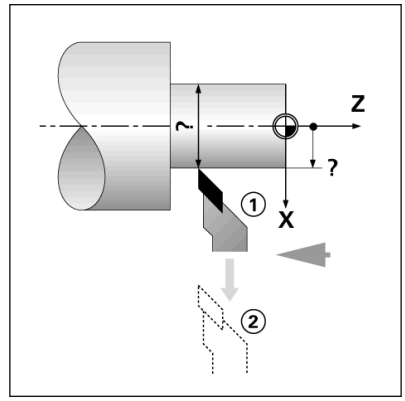
Example: Setting a workpiece datum

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 W550 is in diameter display mode Ø 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.

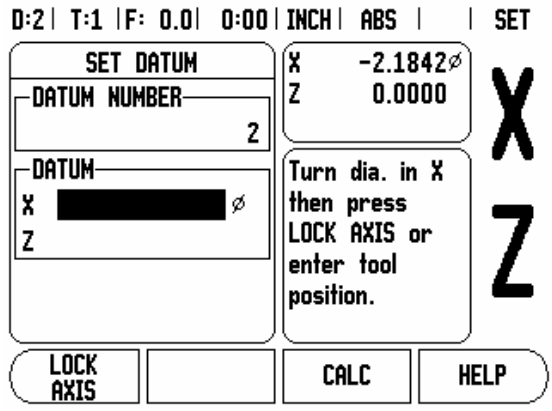


Setting Datums using LOCK AXIS Function

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.



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:

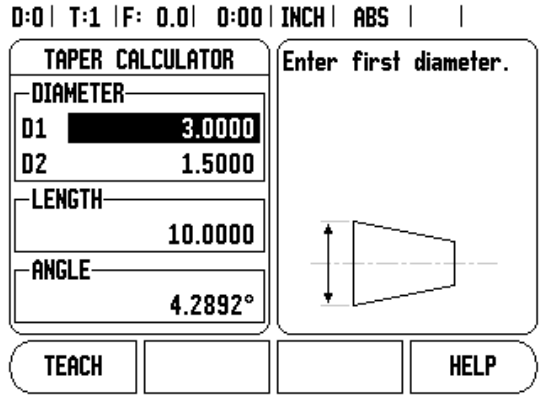
- Length of the taper
- Change in the radius of the taper.

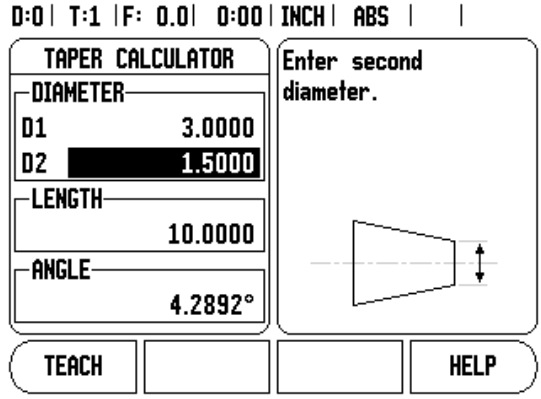
For taper calculation using both diameters (D1, D2) and length requires:

- 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 numeic keys and press enter, or touch the tool to one point and press TEACH.
- Repeat this for the diameter 2 field. When using the TEACH 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

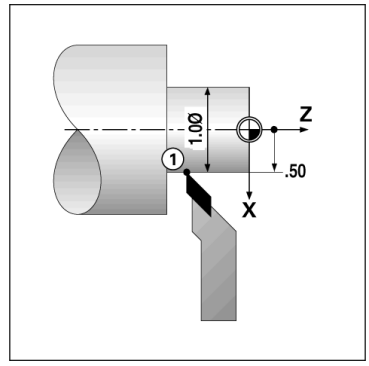
The functionality has been explained previously in this manual (see "Presets" on page 32). The explanation, and examples on those pages are based on a mill application. The basics of those explanations are the same for turning applications with two exceptions; Tool Diameter Offsets (R+/-), and Radius 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 Ø 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. W550 can display either the radius or the diameter for you. When the diameter is being displayed, the diameter symbol Ø is shown next to the position value.



Example:

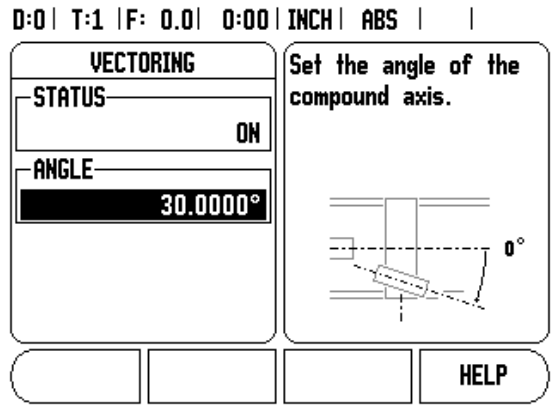
- Radius display, position 1, X =.50
- Diameter display, position 1, X = Ø 1.0

▶ 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".

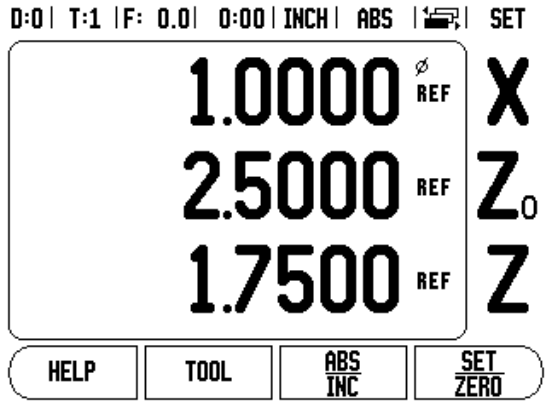
When vectoring is used, the top slide (compound) axis encoder must be assigned to the bottom display axis. The crossfeed component of movement of the axis will then be shown in the top display axis. The longitudinal component of movement of the axis will be shown in the middle display axis.



- 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 W550 Turning application provides a quick method for coupling the Z_0, and Z axes positions on a 3 axis system. The display can be coupled in either the Z_0, and Z displays.



Enabling Z_0, and Z Coupling

To couple the Z_0, and Z axes and have the result displayed on the Z_0 display, press and hold the Z_0 key approximately 2 seconds. The sum of the Z positions will be displayed on the Z_0 display and the Z display will be blanked.

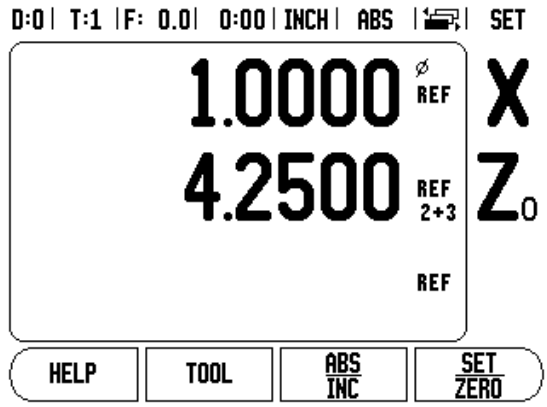
To couple the Z_0, and Z axes and have the result displayed on the Z display, press and hold the Z key for approximately 2 seconds. The sum of the Z positions will be displayed on the Z display and the Z_0 display will be blanked. The coupling is preserved in between power cycles.

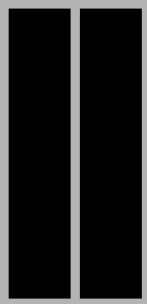
Moving either Z_0, or Z inputs will update the coupled Z position.

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

Disabling Z_0, and Z Coupling

To disable Z Coupling, press the axis key of the display that is blank. The individual Z_0, and Z display positions will be restored.





Technical Information

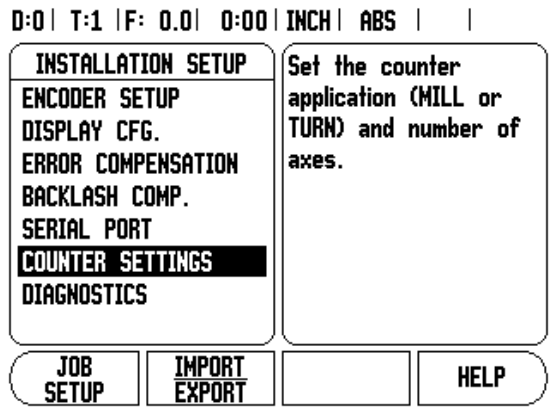
W550

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.

The Installation Setup Data can be imported, or exported using the IMPORT/EXPORT soft key

Exporting the current Installation Setup:

- Press the SETUP soft key.
- Press the INSTALL SETUP soft key.
- Press the IMPORT/EXPORT soft key.
- Press the EXPORT soft key.

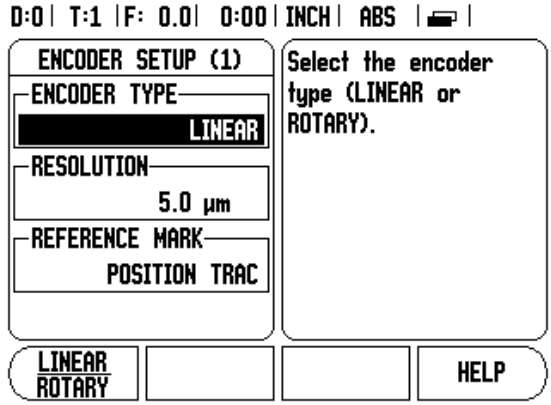
Importing a new Tool Table

- Press the SETUP soft key.
- Press the INSTALL SETUP soft key.
- Press the IMPORT/EXPORT soft key.
- Press the IMPORT soft key.

Encoder Setup

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

- The cursor will default to the ENCODER SETUP field upon opening Installation Setup. Press ENTER. This opens a list of up to 3 possible encoders, labeled INPUT X1, X2, and X3.
- Scroll to the encoder input to be added, or changed, and press ENTER.
- The ENCODER SIGNAL field will automatically be determined.
- 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.



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

- 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. ON allows the DRO to monitor Counting errors. The types of Counting errors are contamination errors (when the signal to the encoder falls below a set limit), and frequency error (when the signal frequency exceeds the set limit).
  - 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 settings, press ENTER then JOB SETUP.

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 W550 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 10^6 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).

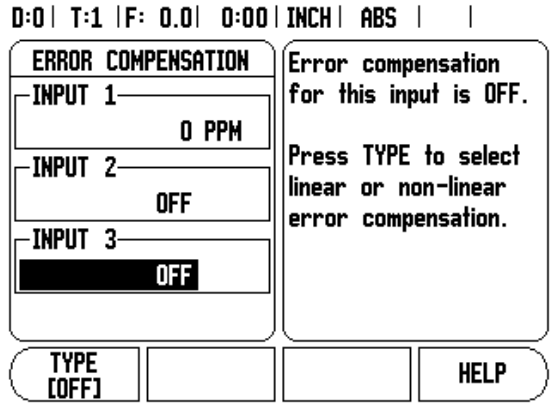
LEC = (500 – 499.95) x 10^6 ppm = 100 ppm (rounded to the nearest whole number).

- 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.

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 W550 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.

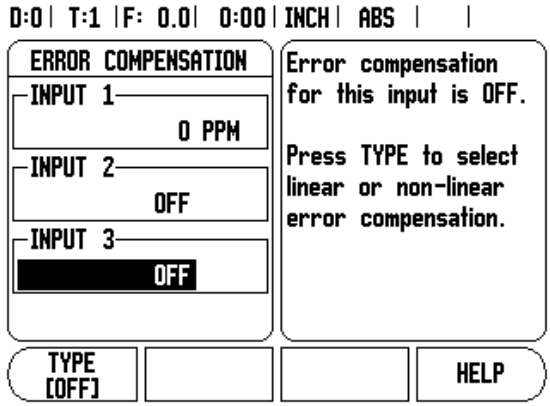


Setup procedure for Non-linear Error

- The encoder has an inherent count direction. This may not reflect the user defined count direction, and is only required for determining the non-linear error compensation.

  To establish the inherent count direction for any installed encoder on a given axis, complete the following:

- ▶ Open the Encoder Setup form, and select the encoder on the axis that is to be addressed.
- ▶ Arrow down to high light the count direction.
- ▶ Use the POSITIVE/NEGATIVE soft key and select Positive. Press enter.
- ▶ Using the C key, return to the main display
- ▶ Move the axis that the encoder is mounted on, and note the direction of movement required for the positive direction.
- The encoder's inherent count direction is now established.



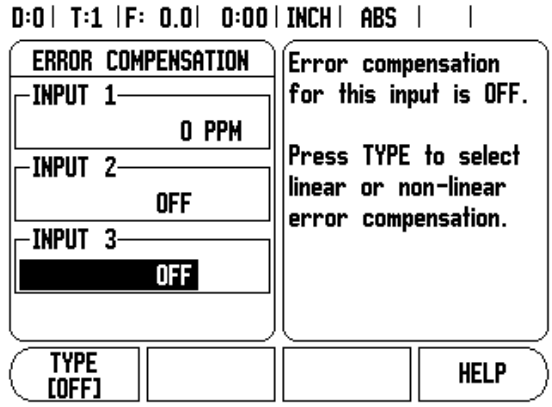


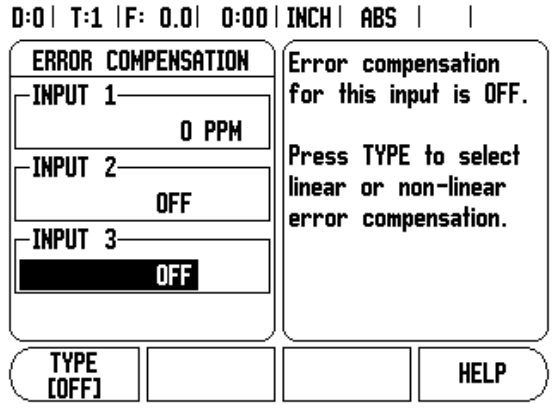
Single reference mark encoders must cross the same reference mark each time the DRO is powered on.

Starting a Non-Linear Error Compensation Table

- Select ERROR COMPENSATION in **Installation Setup**, and 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 ENTER, then press TEACH POSITION.

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. A compensation table may also be imported, or the new table can be exported.

Exporting the current Compensation Table

- Press the IMPORT/EXPORT soft key.
- Press the EXPORT soft key.

Importing a new Tool Table

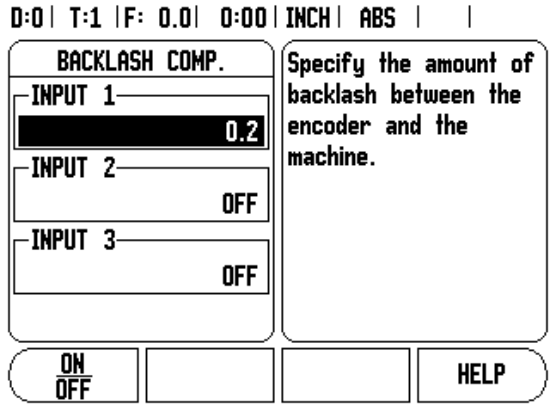
- Press the IMPORT/EXPORT soft key.
- Press the IMPORT soft key.

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.

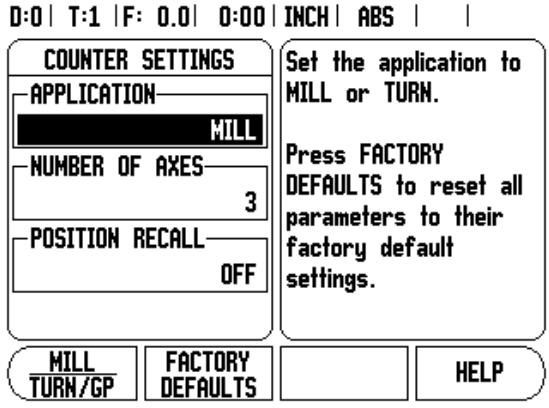


Counter Settings

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.

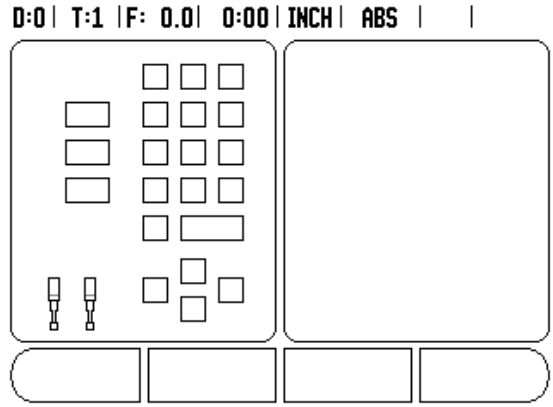


The Position Recall feature, when it is "ON", will store the last position of each axis when power was turned off, and then redisplay that position once power is turned back on.

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 Data Interface

The W550 data interface includes the RS-232 serial port. The serial port supports both bi-directional data communications which allows data to be exported to, or imported from an external device. Also, external operations via the data interface.

Data which can be exported from the W550 to an external serial device includes:

- Job, and Installation configuration parameters.
- Non-linear error compensation tables.
- Measured value output (display values, or probing functions).

Data which can be imported into W550 from an external device includes:

- Remote key commands from an external device
- Job, and installation configuration parameters
- Non-linear error compensation tables

For operations that support data transfer, an IMPORT/EXPORT soft key is available. See "Import/Export" on page 18.

To export, or import data between the W550 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 W550, and the PC is in ASCII text format.

To export data from the W550 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 W550 IMPORT/EXPORT soft key.

To import data into the W550 from a PC, the W550 must first be made ready to receive the data.

- Press the W550 IMPORT/EXPORT soft key.
- After the W550 is ready, setup the terminal communications program on the PC to send the desired file in ASCII text format.



The W550 does not support communication protocols such as Kermit or Xmodem.

Serial Port

The RS-232-C / V.24 data interface serial port is located on the rear panel.

The following devices can be connected to this port:

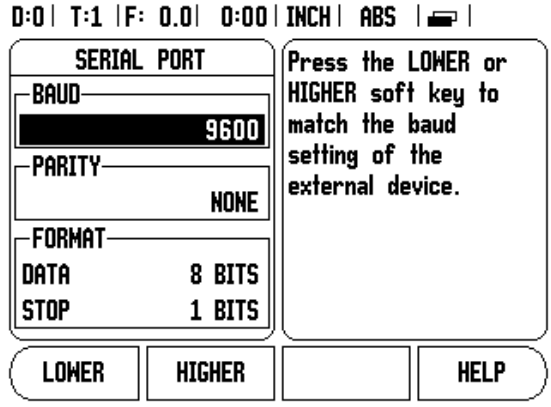
- Printer with serial data interface.
- Personal computer with serial data interface.

A printer, or computer may be connected to the serial port. Part programs, and configuration files may be sent to a printer, or computer. Remote commands, remote key codes, part programs, and configuration files may be received from a computer.

For operations that support data transfer, an IMPORT/EXPORT soft key will be available.

- 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.



II - 3 Installation and Electrical Connections

Installation

The DRO is mounted to a tilt/swivel feature: See "DRO Dimensions" on page 79.

Electrical requirements

Voltage	100 - 240 Vac
Power	30 VA max.
Frequency	50/60 Hz (+/- 3Hz)

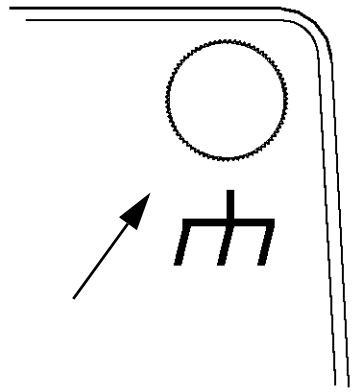
Fuse 630 mA/250 Vac, 5 mm x 20 mm, Slo-Blo (line and neutral fused)

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

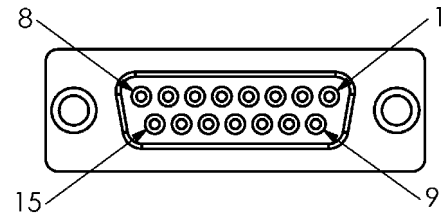
No special preventative maintenance is necessary. For cleaning, wipe lightly with a dry lint-free cloth.

II - 4 I/O Connections

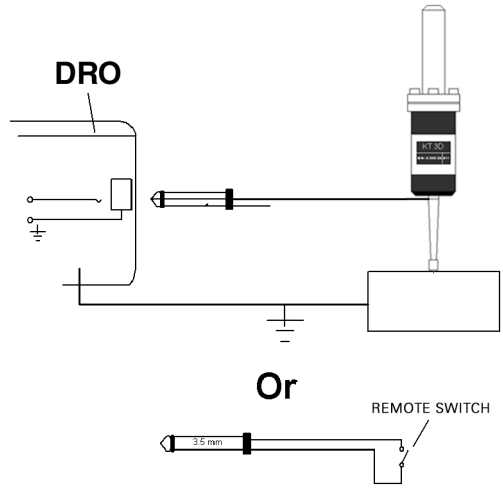
Encoders are plugged into connectors marked Inputs 1, 2, 3, & 4.

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



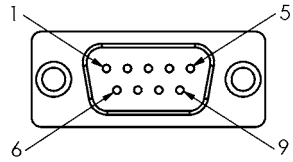
15 Pin receptacle connector for Edge Finder.



Grounding Edge Finder / Remote Switch

Wiring the Serial communication cable

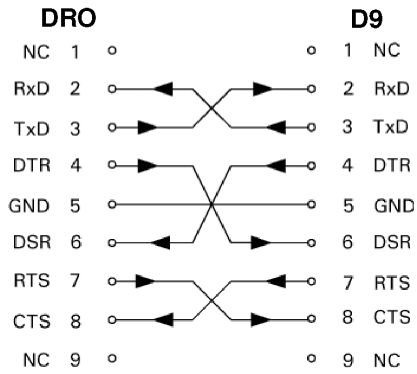
The wiring of the serial communication cable depends on the device being connected (see technical documentation for external device)



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

Pin assignment

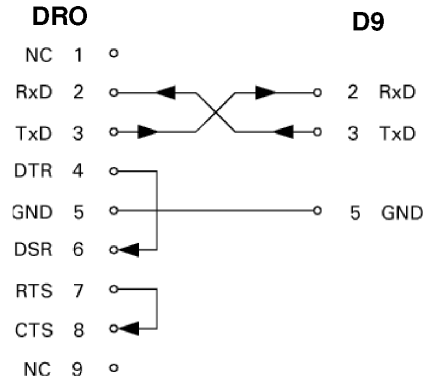
Pin	Assignment	Function
1	No assignment	
3	TXD	- Transmitted Data
2	RXD	- Received data
7	RTS	- Request to send
8	CTS	- Clear to send
6	DSR	- Data set ready
5	SIGNAL GND	- Signal ground
4	DTR	- Data terminal ready
9	No assignment	



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		:	-	3674	.	4498		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 **L**ine **X** axis)

Distance between the probed edges DST = 2853.0012 mm (**D**i**st**ance)

CLX	:	+	3476	.	9963		R	<CR>	<LF>
DST	:		2853	.	0012		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 Error Messages

The following table provides a complete list of error messages that may be received from the W550 DRO.

An explanation of each error message is provided in the following table.

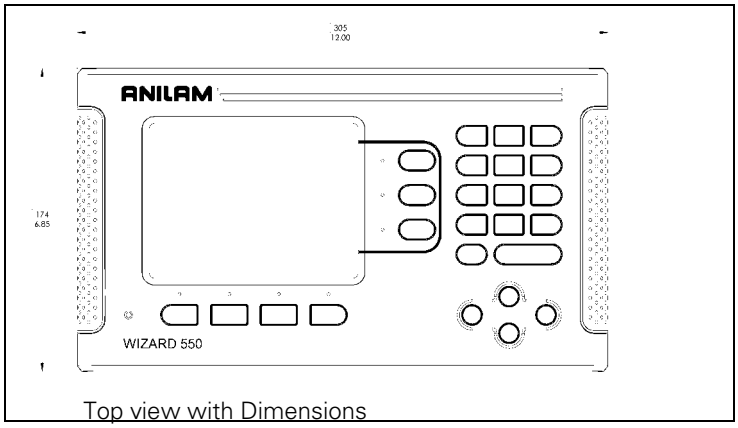
Error	DRO Error Message	Explanation
1	**Power was off.** Press any key to continue.	W550 power was just cycled.
2	**Contamination & Frequency Error:** Press C to clear error.	Contamination, and frequency errors occurred at corresponding encoder. User intervention to monitor, and/or correct the encoder.
3	**Contamination Error:** Press C to clear error.	Contamination error occurred at corresponding encoder. User intervention to monitor, and/or correct the encoder.
4	**Frequency error:** Press C to clear error.	Frequency error occurred at corresponding encoder. User intervention to monitor, and/or correct the encoder.
5	**Miscount error:** Press C to clear error.	Miscount error occurred at corresponding encoder. User intervention to monitor, and/or correct the encoder.
6	**Display Overflow error:** Move within display range.	The encoder is currently out of the display range specified by the user. Move the encoder to within the display range, or modify the encoder display.
7	**Error:** The valid range for number of holes is 1-99.	The number of holes defined for the current pattern is out of range. User must adjust the number of holes.
8	**Error:** The radius must be greater than 0.0.	The user defined radius must be greater than zero to define a circle.
9	**Error:** The spacing must be greater than 0.0.	The spacing between holes in a pattern must be greater than zero to define a pattern.
10	**Error:** The start and end points can not be the same.	The start, and end points of the field must be unique to define a line.
11	**Error:** The distance from the center is not equal to the radius.	The distance from the center value is invalid, and must be modified by the user.
12	**Error:** The ARCCOS function can only operate on values from - 1to 1.	Range error on the value to fetch the arccos of.

Error	DRO Error Message	Explanation
13	**Error:** The ARCSIN function can only operate on values from - 1to 1.	Range error on the value to fetch the arcsin of.
14	**Error:** The TAN function is undefined at 90, and -90 degrees.	Range error on the value to fetch the tangent of.
15	**Error:** The square root of a negative number is undefined.	Can not take the square root of a negative number.
16	**Error:** Divide by zero error.	Can not divide by zero.
17	Unable to transfer configuration parameters. The external device is not ready.	The external device hooked to the serial port is not working. Check the physical connection, and verify that the serial communications parameters are correct.
18	**Error:** One or more of the configuration parameters was invalid, and could not be loaded.	The configuration file being imported is corrupt. Import file again. If error persists, import a different configuration file.
19	**Error:** Parameter was invalid. It has been set to it's previous value.	An invalid configuration parameter was discovered on import, and was reset to the previous value. Check the configuration file that failed, and import again.
20	**Error:** The valid range for scale factor is -10 to -0.1, and 0.1 to 10.	The current scale factor value is out of range. Adjust the scale factor value to reflect a valid number.
21	**Error:** At least one axis must be selected for DRO 1.	When defining multiple DRO, and corresponding views, user must define at least one axis for the default DRO 1.
22	**Error:** The valid range for the error correction factor in parts per million is -99999 to +99999.	Out of range for the error correction factor defined within error compensation. When defining the error correction factor, the user must use the correct range.

II - 7 Dimensions

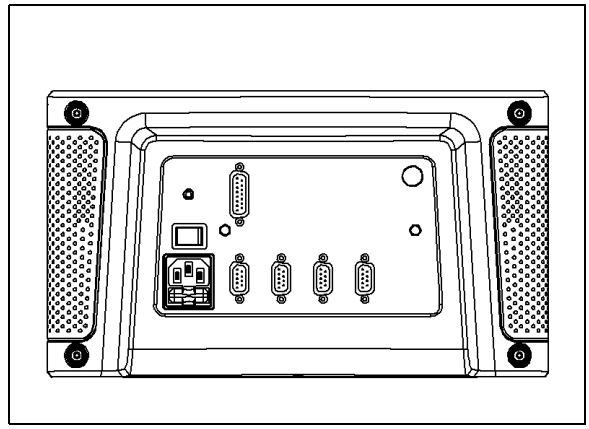
DRO Dimensions

Dimensions in inches/mm

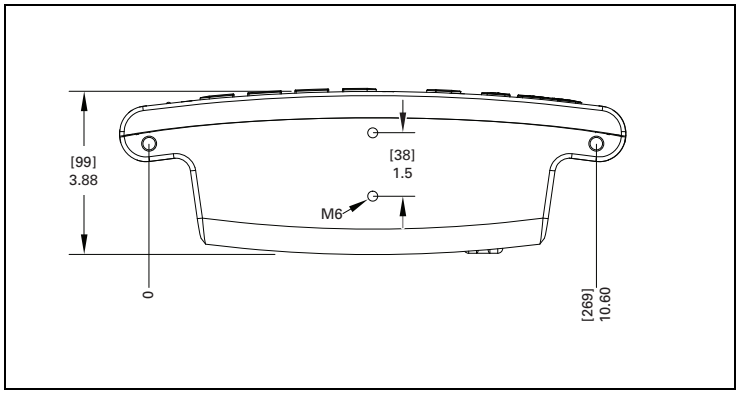


Top view with Dimensions

Front view with Dimensions



Back view



Bottom view with Dimensions

Numerics
1/2 soft key 35

A
Absolute 12
absolute coordinates 3
Absolute Distance Preset 32
Absolute Workpiece Positions 3
Actual Position, Nominal Position, and Distance-To-Go 2
Actual Value/Distance-To-Go soft key 12
Arc Milling 44
Automatic Non Linear error 63
Axis Labels 6

B
Backlash Compensation 65

C
Calc key ii
Calc soft key 19
Calling a Tool from the Tool Table (Turning) 48
Circle and Linear Pattern 37
Console adjustment 18
Counter Settings 65

D
Data Input Forms 9
Datum soft key (Milling) 27
Datum Soft Key (Turning) 48
Datums 2
Diagnostics 66
Diameter Axes (Milling) 14
Disable Ref soft key 11
Display Area 6
Display configuration 59
DRO Dimensions 79

E
Edge Finder (milling applications only 14
Electrical Requirements 69
Enable Ref soft key 11
Enable/Disable Ref Function 11
Encoder Reference Marks 5
Encoder Setup 57
Environmental Specifications 69
Error Compensation 59
Error Messages 9, 77

F
Features 36
features 36
Fundamentals of Positioning 2

G
General Navigation 7
Graphic Positioning Aid 7
Grounding (Earth) 69

H
Help Screen 8, 28, 29

I
Import/Export (setting) 18, 22
Incline & Arc Milling 41
incremental coordinates 3
Incremental Distance Preset 34
Incremental workpiece positions 3
Index soft key 8
Installation Setup Parameters 56
Instruction Box 9

J
Job Clock 15
Job Setup menu 12
Job Setup Parameters 13

L
Language (setting) 18
Linear Error Compensation 60

M
Measured Value Output 15, 72
Mirror 14

N
Near Zero Warning 15
No Ref soft key 10

O
Operating Instructions 1
Operating Modes 7, 12

P
Position Recall 66
Power Up 10
Preset 32
Preventative Maintenance 69
Probing Functions 27

R
Radius/Diameter 51
Reading head position 4
Ref Symbols 6
Reference Mark 10
Remote Switch 17
RPM Calculator (Milling) 20

S
Scale factor 13
Screen Layout 6
Serial Port 68
Set Zero soft key 19
Setting Tool Offsets 46
Setup 12
Setup proceedure for Non-linear Error 61
Setup soft key 12
Skew Compensation 16
Soft key Labels 6
Status Bar 15
Status Bar Settings 15

T
Taper Calculator 50
Tool data entering 24
Tool Soft Key 21
Tool Table 21
Tool Table (Turning) 46
Tool Table Usage 24

U
Units of measurement, setting 13

V
Vectoring 52

Z
Z Coupling 52
Zero Angle Reference Axis 4

Index

HEIDENHAIN CORPORATION

333 East State Parkway
Schaumburg, IL 60173-5337 USA
☎ +1 (847) 490-1191
FAX +1 (847) 490-3931
E-Mail: info@heidenhain.com
www.heidenhain.com

ISO 9001
CERTIFIED

CE

532841-24 Ver00 10/2009

AMI
Automation and Metrology
1-888-354-2525
auto-met.com