

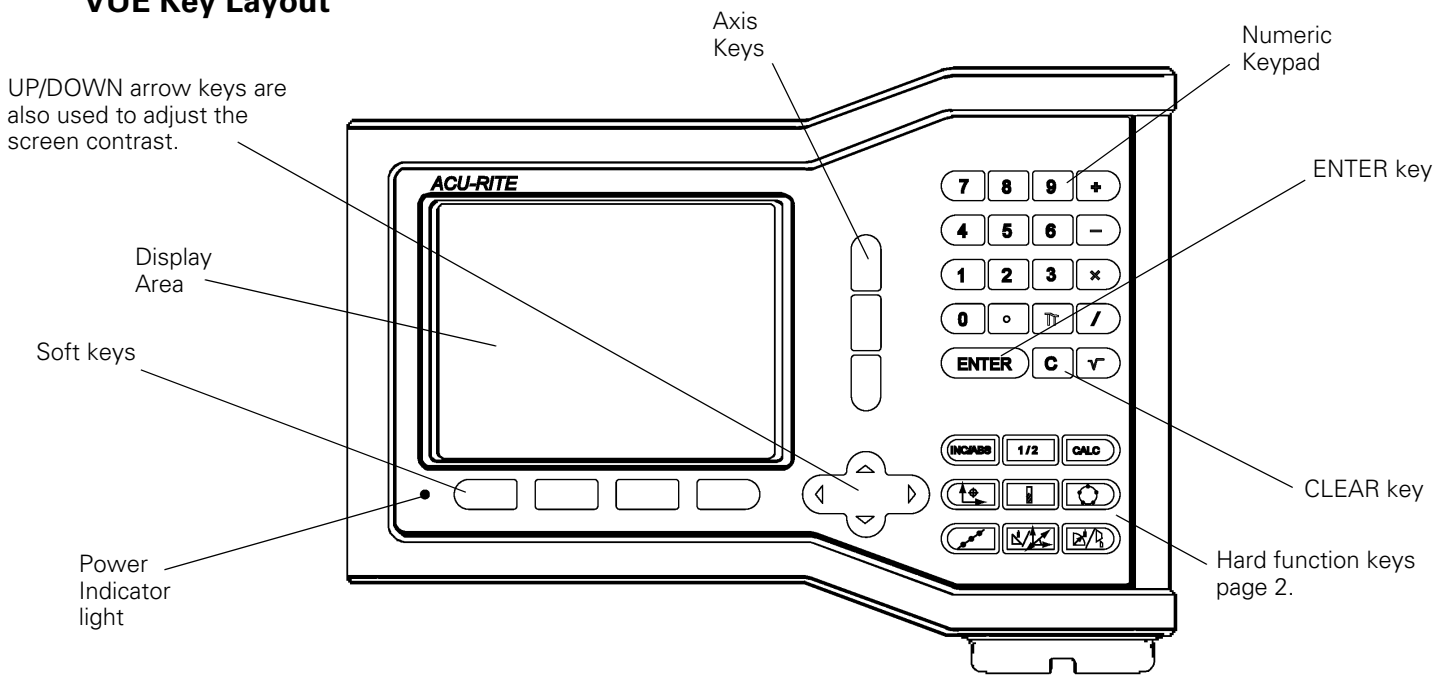
VUE READOUTS



ACU-RITE[®]

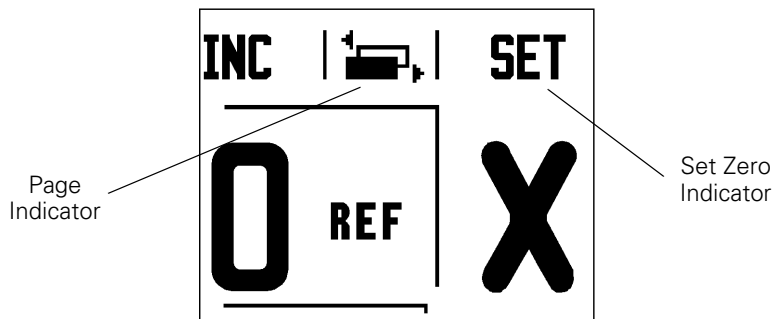
REFERENCE MANUAL

VUE Key Layout



VUE Soft Keys

There are two pages of soft key functions to select from the operating modes. Use the LEFT/RIGHT arrow keys 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.



Soft Key function	Soft key Symbol
Opens the Job Setup menu and provides access to the Installation Setup soft key (page 4).	SETUP
Press when ready to identify a reference mark (page 4).	ENABLE REF
Opens the Tool Table. See page 7 for Milling, and page 20 for Turning. <i>The TOOL key is a soft key for the one axis readout only.</i>	TOOL
Opens the Datum form to set the datum for each axis. See page 8 for Milling, and page 22 for Turning. <i>The DATUM key is a soft key for the one axis readout only.</i>	DATUM

Soft Key function	Soft key Symbol
Opens on-screen HELP instructions.	HELP
Toggles between inch and millimeter units (page 4 under Units).	INCH MM
This soft key toggles between radius and diameter displays. This function is for Turning applications only (page 24).	RAD DIA
Toggles between SET/ZERO functions. Used with individual axis keys (page 6).	SET ZERO

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

Refer to the Setup section. Begin by pressing the **SETUP** key, followed by pressing the **INSTALLATION SETUP** soft key. Press the access code numbers **8** **8** **9** **1** using the numeric key pad, then press the **ENTER** key. The readout is now ready for machine parameter setting operations.

IMPORTANT
Supervisors may wish to remove this page from the Reference Manual after initially setting up the readout system. Retain in a safe place for future use.

Introduction

Software Version

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



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

VUE

DRO axis availability.



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



Warning

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



Caution - Risk of electric shock

e.g. when opening a housing.

VUE Fonts

The chart below shows how the different variables (soft keys, hard keys) are represented within the text of this manual:

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

Hassle-Free Warranty

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

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

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

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

Table of Contents

I – 1 Introduction to VUE

Layout of Screen.....	1
VUE Hard Keys	2
Operating Modes.....	3
Reference Mark Evaluation.....	3
ENABLE/DISABLE REF function.....	4
Job Setup Parameters	4
Units.....	4
Scale Factor	4
Mirror	4
Diameter Axes	5
Near Zero Warning.....	5
Status Bar Settings	5
Job Clock	5
Console Adjustment	5
Language	6
Import/Export.....	6
Set/Zero Soft Key Details.....	6

I – 2 Milling Specific Operations

Key Functions Detailed	7
Tool Hard Key.....	7
Tool Table.....	7
Tool Compensation	7
Calling the Tool from the Tool Table	8
Datum Setting.....	8
Datum Setting with a Tool	9
Presets.....	10
Absolute Distance Preset	10
Incremental Distance Preset.....	12
1/2 Hard Key	13
Circle and Linear Pattern Hard Keys	13
Functions for milling patterns	13
Linear Pattern.....	15
Incline & Arc Milling.....	15
Incline Milling.....	16
Arc Milling.....	18

I – 3 Turning Specific Operations

Tool Table.....	20
Tool Display Icon.....	20
Datum Setting.....	22

Taper Calculator Hard Key	23
Presets.....	24
Radius/Diameter Soft Key.....	24
Vectoring.....	24

II – 1 Installation Setup

Installation Setup Parameters	25
Encoder Setup	25
Display Configuration	26
Coupling.....	26
Z Coupling (turning applications only)	26
Enabling Z Coupling	27
Disabling Z Coupling	27
<i>Error Compensation</i>	27
<i>Linear Error Compensation</i>	28
<i>Non-Linear Error Compensation</i>	28
Backlash Compensation	29
Counter Settings.....	30
Diagnostics	30
Keypad Test	30
Display Test	30

II – 2 Installation and Electrical Connections

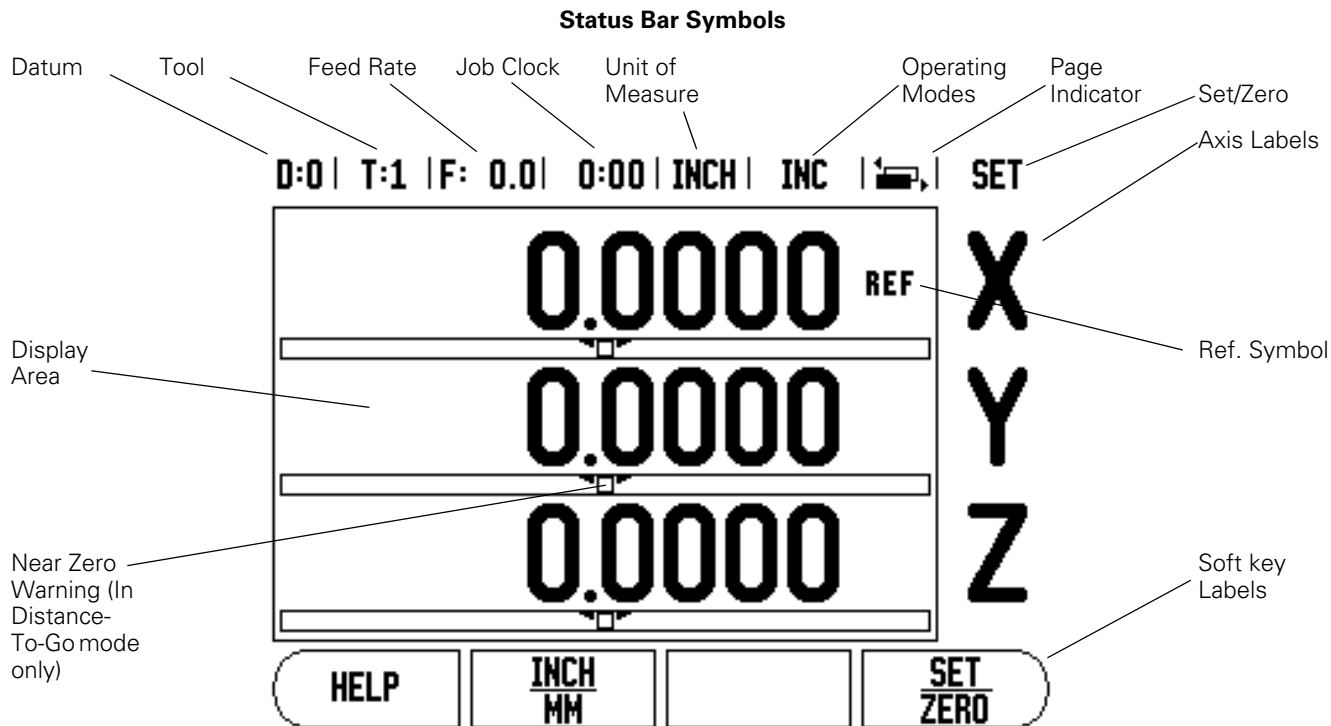
Installation.....	31
Electrical requirements	31
Environmental.....	31
Preventative maintenance	31

II – 3 Dimensions

Accessory ID Number.....	32
DRO mounting with arm (reference information)	33
DRO mounting with base	33

I – 1 Introduction to VUE

Layout of Screen




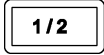
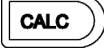
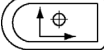
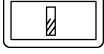
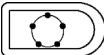

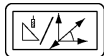

ACU-RITE's VUE readout provides application-specific features that allows you to obtain the most productivity from your manual machine tools.

- **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.

VUE Hard Keys

Hard keys on the VUE readouts vary depending on the number of axis the readout includes. A one axis readout will only have the first three hard keys shown below. A two, or three axis readout will have all the hard keys shown below.

The last two keys are specific to a VUE readout that is either for milling or turning. The first symbol applies to a milling readout, and the second symbol applies to a turning readout.

Hard Key function	Hard Key Symbol
Switches display between operating modes: Distance-To-Go (Incremental) / Actual Value (Absolute) (see page 3).	
Used to divide the current position by two (see page 13).	
The CALC hard key opens the Calculator functions for standard math, trigonometry, RPM and Taper (Turning only) functions (see page 23).	
The DATUM hard key opens the Datum form to set the datum for each axis (for Milling see page 8, for Turning see page 22).	
The T00L hard key opens the Tool Table (for Milling see page 7, for Turning see page 20).	
The CIRCLE PATTERN hard key (Mill only) opens the Circle Pattern form. This calculates the hole positions (page 13).	
The LINEAR PATTERN hard key (Mill only) opens the Linear Pattern form. This calculates the hole positions (page 13).	
The INCLINE MILLING hard key (Mill) or the VECTORING hard key (Turn) opens the Incline milling forms (page 16), or the Vectoring form (page 24).	
The ARC MILLING hard key (Mill) or the TAPER CALC hard key (Turn) opens the Arc milling forms (page 18), or the Taper Calc form (page 23).	

Operating Modes

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

While in the **ABSOLUTE (Actual Value)** mode, if the VUE is configured for Milling applications, only the tool length offsets are active. Both the diameter and length offsets are used in the **INCREMENTAL (Distance-To-Go)** mode to calculate the amount of “distance-to-go” required to get to the desired nominal position relative to the edge of the tool that will be doing the cutting.

If the VUE is configured for turning, all tool offsets are used in both the **ABSOLUTE** and **INCREMENTAL** modes.

Press the **ABS/INC** hard key (see Fig. I.1) to toggle between these two modes. To view other soft key functions in either **ABSOLUTE** or **INCREMENTAL** mode, use the **LEFT/RIGHT** arrow keys.

The turning application provides a quick method for coupling the Z axes position on a 3 axis system. For more information, see “Coupling” on page 26.

Reference Mark Evaluation

The VUE reference mark evaluation feature automatically re-establishes the relationship between axis slide positions and display values that you last defined by setting the datum.

For each axis with an encoder that has reference marks the **REF** indicator will flash for that axis (see Fig. I.2). After crossing over the reference marks, the indicator will stop flashing and change to a non-flashing “REF”.

Working without reference mark evaluation

You can also use the VUE without crossing over the reference marks. Press the **NO REF** soft key to exit the reference mark evaluation routine and continue.

You can still cross over reference marks at a later time 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.



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.



Fig. I.1 Distance-To-Go (INCREMENTAL) / Actual Value (ABSOLUTE) Hard key

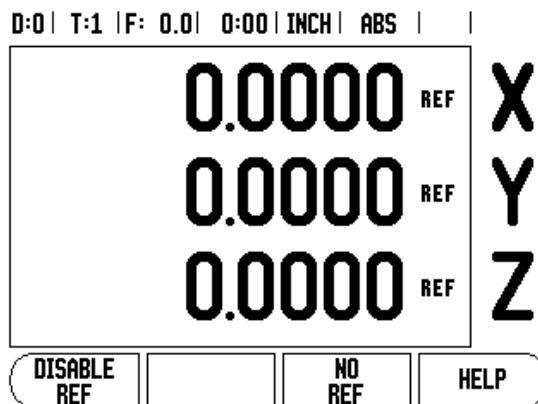


Fig. I.2 Screen for establishing Ref Marks

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.

You do not have to cross over the reference marks of all the encoders, only those that you need. Once reference marks for all desired axes are established, press **NO REF** soft key to cancel out of routine. If all reference marks have been found, the VUE will return to the DRO display screen automatically.

Job Setup Parameters

To view and change Job Setup parameters, first press the **SETUP** soft key, then use the Up/Down arrow keys to highlight the parameters of interest and press the ENTER key. See Fig Fig. 1.3.

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.

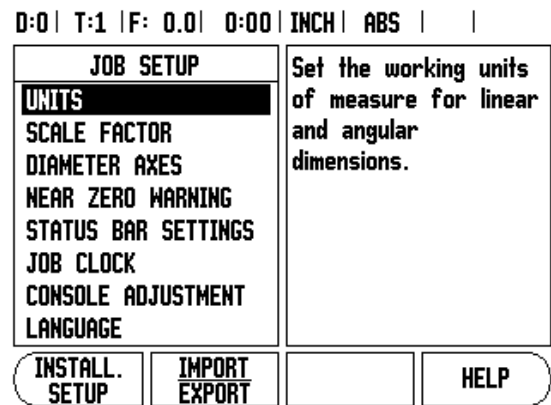


Fig. 1.3 Job Setup screen in milling

Diameter Axes

Select Diameter Axes 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. See Fig. I.4. For turning applications see "Radius/Diameter Soft Key" on page 24 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**.

Near Zero Warning

The Near Zero Warning form is used to configure the bar graph that 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 simply 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.

Console Adjustment

The LCD's 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 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 switches to screen saver mode. The idle time may be set from 30 to 120 minutes. The display saver can be disabled during the current power cycle.

D:0 T:1 F: 0.0 0:00 INCH ABS								
DIAMETER AXES	Set to ON to display the position as a diameter value.							
<table border="1"> <tr> <td colspan="2" style="text-align: center;">DIAMETER AXES</td> </tr> <tr> <td>X</td> <td style="text-align: right;">OFF</td> </tr> <tr> <td>Y</td> <td style="text-align: right;">OFF</td> </tr> <tr> <td>Z</td> <td style="text-align: right;">OFF</td> </tr> </table>		DIAMETER AXES		X	OFF	Y	OFF	Z
DIAMETER AXES								
X	OFF							
Y	OFF							
Z	OFF							
ON/OFF	HELP							

Fig. I.4 Diameter Axes form

Language

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

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

Import/Export

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

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

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.

When the state is Set, see Fig. I.5, and the VUE is in Actual Value mode, selecting an Axis key opens the Datum form for the selected axis. If the VUE is in Distance-To-Go mode, a Preset form opens.

When the state is Zero, and the VUE 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 VUE 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.

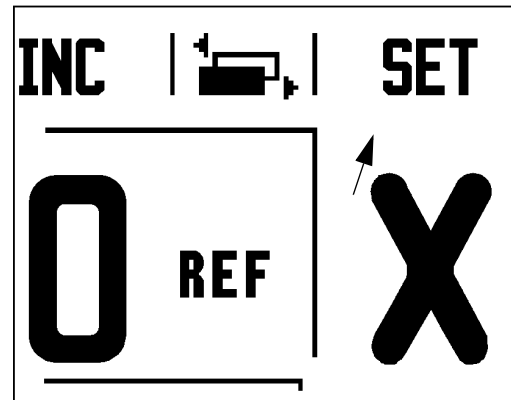


Fig. I.5 Set Zero Indicator

I – 2 Milling Specific Operations

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

Key Functions Detailed

Tool Hard Key


This Hard 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 VUE can store up to 16 tools within the tool table.

Tool Table

The VUE tool table provides a convenient way to store diameter and length offset information for each of the tools you commonly use. You can enter up to 16 tools. (See Fig. I.6 Tool Table in Milling).

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

Function	Soft key
This key allows the operator 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. Only available in TOOL LENGTH field.	TEACH LENGTH
This will open the TOOL TYPES form for selection. Only available in TYPE field.	TOOL TYPES

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

TOOL TABLE (DIA/LENGTH)		
1	2.000/	20.000 MM PILOT DRL
2	5.000/	14.000 MM ROUGH ML
3	25.000/	50.000 MM FL END ML
4	6.000/	12.000 MM BROACH
5	10.000/	25.000 MM ROUGH ML
6	2.000/	0.000 MM FL END ML
7	2.500/	0.000 MM FL END ML
8	3.000/	5.000 MM

TOOL AXIS [Z] CLEAR TOOL USE TOOL HELP

Fig. I.6 Tool Table in Milling

Tool Compensation

VUE has tool compensation. This allows you to enter workpiece dimensions directly from the drawing. The displayed distance to go is then automatically lengthened (R+) or shortened (R-) by the value of the tool radius. See Fig. I.7. For more information see "Presets" on page 10.

The length offset may be entered as a known value or the VUE 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, see Fig. I.8.

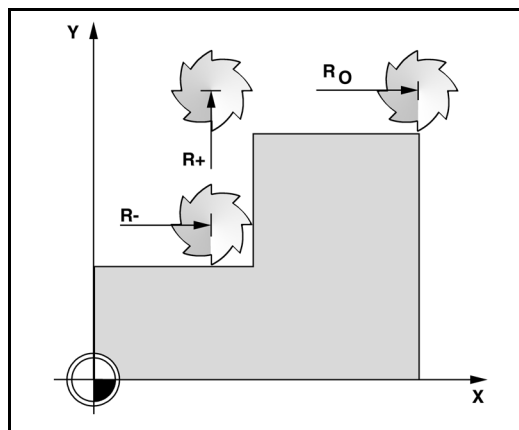


Fig. I.7 Tool compensation

Sign for the length difference ΔL

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

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

As indicated above it is also possible to have the VUE determine a tool's length offset. This method involves touching the tip of each tool to a common reference surface. This allows the VUE 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. The VUE will calculate an offset relative to this surface. Repeat the procedure for each additional tool using the same reference surface.



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

Calling the Tool from the Tool Table

To call a tool, press the T00L hard key. Use the Up/Down arrow keys to cursor through the selection of tools (1-16). Highlight the tool you want. Verify the proper tool has been called, and press the T00L or C key to exit.

Datum Setting

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

Setting datum points is done best by using the VUE probing functions with a tool.

Datum points are set in the conventional manner by touching the edges of the workpiece, one after the other with a tool and manually entering the tool positions as datum points. See Fig. I.9 & Fig. I.10.

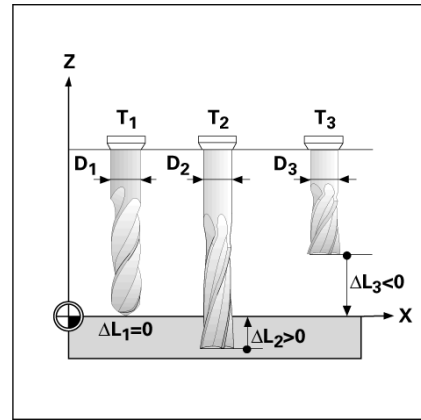


Fig. I.8 Tool length and diameter

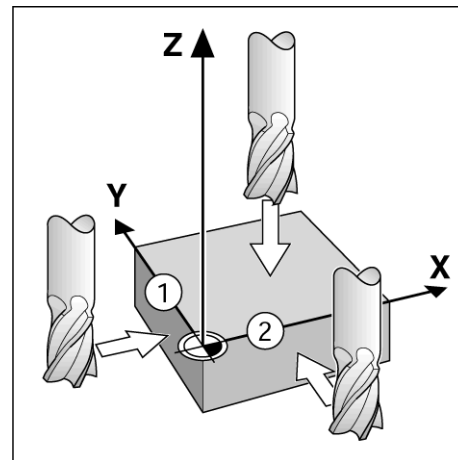


Fig. I.9 Touching the edges

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

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

PROBE [] [] HELP

Fig. I.10 SET DATUM form

Datum Setting with a Tool

Using a tool to set datum points, you can still use VUE probing functions. See Fig. I.11 & Fig. I.12.

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, VUE takes into account the current tool's entered tip diameter.

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

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

Datum axis: X = 0

Tool diameter D = 0.25"

Press DATUM hard key.

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.

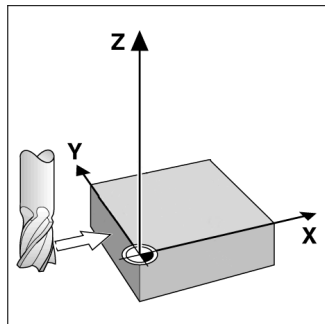


Fig. I.11 Setting datum using an edge

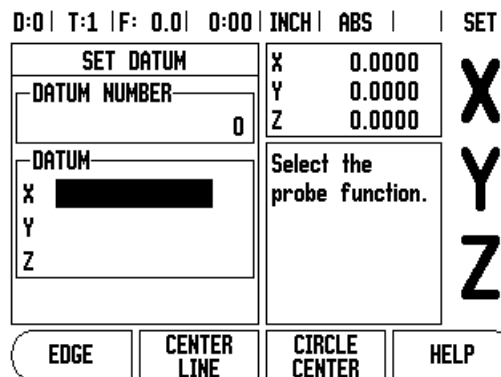


Fig. I.12 Set Datum screen

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. See Fig. I.13.

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. See Fig. I.14 & Fig. I.15.

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



Pressing an axis key recalls the last entered preset value for that axis.

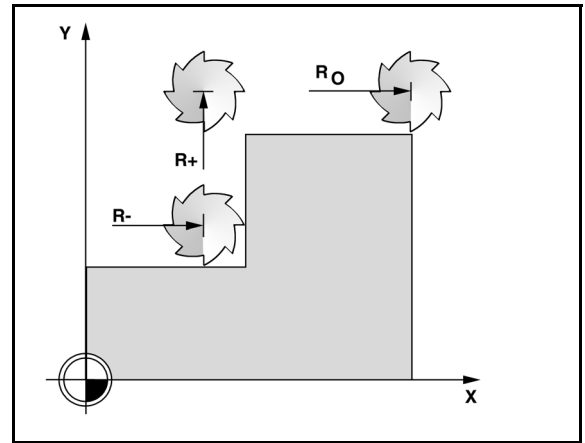


Fig. I.13 Tool radius compensation

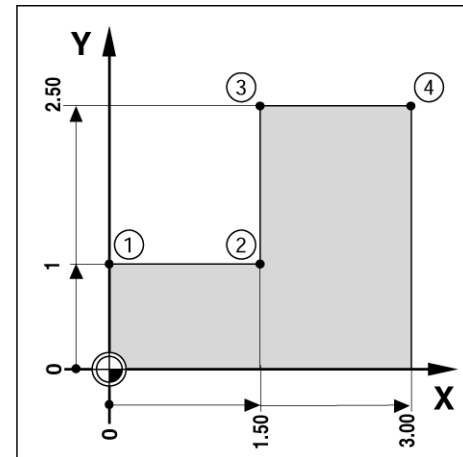


Fig. I.14 Single cycle preset

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 **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 **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.

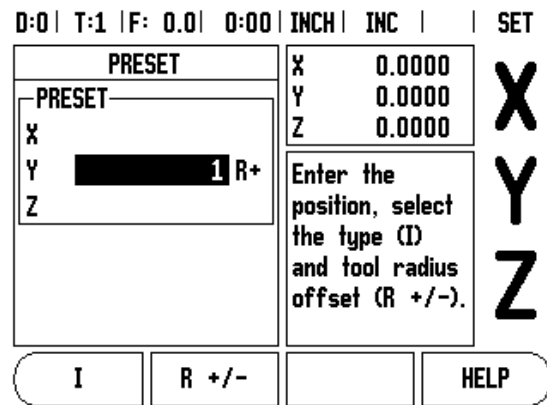


Fig. I.15 Preset screen

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. See Fig. I.16 & Fig. I.17.

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 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 Y axis key.

Enter nominal position value for hole 1: $Y = 1''$.

Ensure no tool radius compensation is showing.

Press the Z axis 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 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 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.

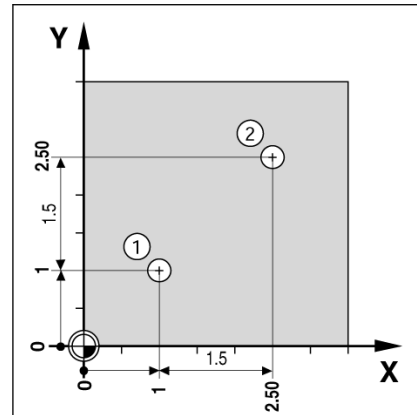


Fig. I.16 Drilling example

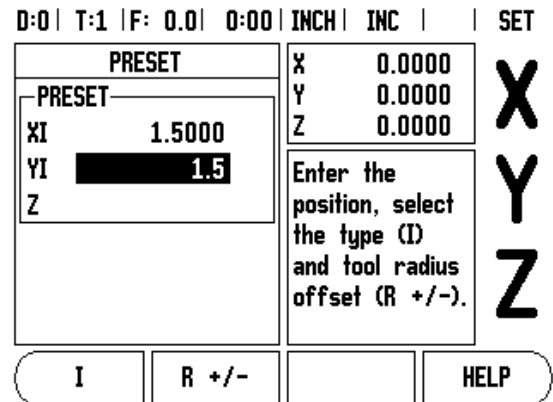


Fig. I.17 Incremental screen

1/2 Hard Key

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



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





Circle and Linear Pattern Hard Keys

This section describes the hole pattern functions for Circle and Linear patterns.

Press the CIRCLE or LINEAR PATTERN hard key to access the Pattern function then enter the required data. VUE then calculates the positions of all the holes and displays the pattern graphically on the screen.

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

Functions for milling patterns

Function	Soft key
Press this to see the layout of the current pattern.	
Press to go to previous hole.	
Press to manually advance to the next hole.	
Press to end drilling.	

Example: Enter data and execute a circle pattern. See Fig. I.18 & Fig. I.19.

Holes (no. of): 4
 Coordinates of center: X = 2.0" / Y = 1.5"
 Bolt circle radius: 5
 Start angle: Angle between X axis and first hole: 25°
 Hole depth: Z = -0.25"

1st step: Enter data

Press **CIRCLE PATTERN** hard key.
 Enter the type of circle pattern (full). Cursor to the next field.
 Enter the number of holes (4).
 Enter the X and Y coordinates of the circle center (X=2.0), (Y=1.5).
 Cursor to the next field.
 Enter the radius of the circle pattern (5).
 Enter the start angle (25°).
 Enter the end angle (295°) (this can only be changed if entering a "segment"). The **END ANGLE** is defined as the angle from the positive X-axis to the end of the pattern.

Enter the depth when needed. The depth of the hole is optional and may be left blank. If not required, press **ENTER**.

Three views are available: Incremental DRO, Pattern Graphic, and Absolute DRO. Press the **VIEW** soft key to toggle through the available screens.

2nd step: Drill

Move to hole:

Traverse the X and Y axes until display value zero.

Drill:

Traverse to display value zero in the tool axis. After drilling, retract the drill in tool axis.

Press the **NEXT HOLE** soft key.

Continue to drill the remaining holes in the same way.

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

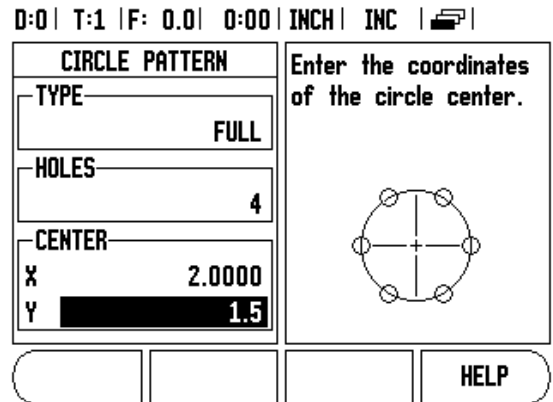


Fig. I.18 Beginning of Circle Pattern form

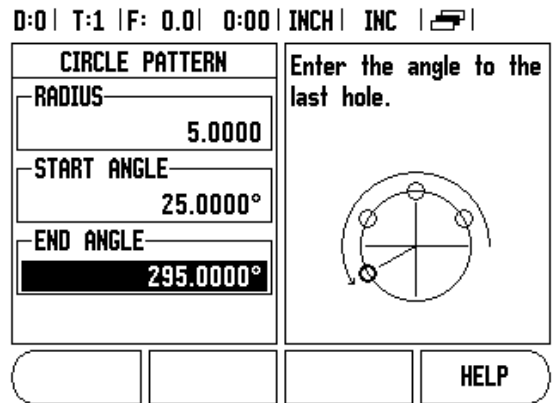


Fig. I.19 Page 2 of Circle Pattern Form

Linear Pattern

Information required (See Fig. I.20 Linear Pattern Example):

- Linear pattern type (array or frame)
- First hole (1st hole of the pattern)
- Holes per row (number of holes in each row of pattern)
- Hole spacing (the spacing or offset between each hole in the row)
- Angle (the angle or rotation of the pattern)
- Depth (the target depth for drilling in the tool axis)
- Number of rows (number of rows in the pattern)
- Row spacing (the spacing between each row of the pattern)

Information entry and operation of the Linear Pattern feature is very similar to the Hole Pattern feature described earlier.

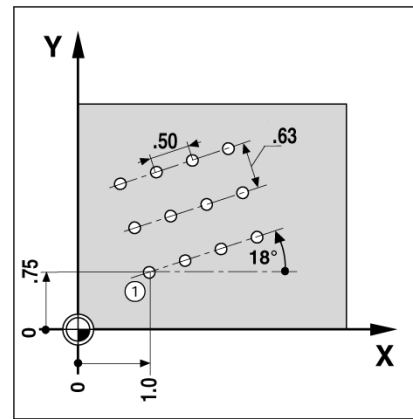


Fig. I.20 Linear Pattern Example

Incline & Arc Milling

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

The following soft keys are also available while in the Entry Form.

Function	Soft key
Press this to select a plane.	PLANE [XY]
Press to execute the milling operation.	RUN
Press this to use the existing position.	TEACH
Press to return to the previous step.	PREVIOUS STEP
Press to advance to the next step.	NEXT STEP

Incline Milling

Entry Form: (See Fig. I.21 Entry Form: Start Point) and (Fig. I.22 Entry Form: End Point)

The Incline Milling form is used to specify the flat surface to be milled. Press the **INCLINE MILLING** hard key to open the form

- **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 **ENTER** or **RUN** to execute the surface milling operation. Press **C** to exit the form without executing. Settings are retained until power is turned off.

Execution

- Execute the milling operation by opening the entry form and pressing the **RUN** soft key or **ENTER** key. The screen switches to the incremental DRO view.
- Initially, the DRO shows the current incremental moving distance from the start point. 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.
- After pressing **NEXT PASS**, the incremental display shows the distance from the next step along the line's contour.
- If no step size was specified, the incremental display always shows the distance from the closest point on the line. To follow the contour, move the two axes in small steps, keeping the (X, Y) positions as close to 0 as possible.

D:0 T:1 F: 0.0 0:00 INCH ABS	
MILL LINE	
PLANE	XY
START POINT	
X	0.0000
Y	0.0000
PLANE [XY]	RUN HELP

Fig. I.21 Entry Form: Start Point

D:0 T:1 F: 0.0 0:00 INCH ABS	
MILL LINE	
END POINT	
X	0.0000
Y	0.0000
STEP	
0.0000	
TEACH	RUN HELP

Fig. I.22 Entry Form: End Point

- When executing a surface milling operation, three views are available: incremental DRO, contour, and absolute DRO. Press the **VIEW** soft key to toggle through the available screens.
- The contour view shows the position of the tool relative to the milling surface. When the crosshair representing the tool is on the line representing the surface, the tool is in position. The tool crosshair remains fixed in the center of the graph. As the table is moved, the surface line moves.
- Press the **END** soft key to exit the milling operation.



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.



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.

Arc Milling

Entry Form: (See Fig. I.23 Entry Form: Center Point), (Fig. I.24 Entry Form: Start Point) and (Fig. I.25 Entry Form: Radius)

The Arc Milling form is used to specify a curved surface to be milled. Press the ARC MILLING hard key to open the form.

- **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.
- **Start Point:** Enter the coordinates of the start point.
- **End Point:** Enter the coordinates of the end point.
- **Radius:** Enter the radius of the arc.
- **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 ENTER or **RUN** to execute the milling operation. Press C to exit the form without executing. Settings are retained until power is turned off.

Execution

- Execute the milling operation by opening the entry form and pressing the **RUN** soft key or ENTER key. The screen switches to the incremental DRO view.
- Initially, the DRO shows the current incremental distance from the start point. 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.
- After pressing **NEXT PASS**, the incremental display shows the distance from the next step along the arc's contour.
- If no step size was specified, the incremental display always shows the distance from the closest point on the arc. To follow the contour, move the two axes in small steps, keeping the (X, Y) positions as close to 0 as possible.
- When executing a surface milling operation, three views are available: incremental DRO, contour, and absolute DRO. Press the **VIEW** soft key to toggle through the available screens.

D:0 T:1 F: 0.0 0:00 INCH ABS	
MILL ARC	
PLANE	XY
CENTER POINT	
X	0.0000
Y	0.0000
PLANE [XY]	RUN HELP

Press PLANE to select the plane.

Fig. I.23 Entry Form: Center Point

D:0 T:1 F: 0.0 0:00 INCH ABS	
MILL ARC	
START POINT	
X	0.0000
Y	0.0000
END POINT	
X	0.0000
Y	0.0000
TEACH	RUN HELP

Enter the incline mill arc FROM point.

Fig. I.24 Entry Form: Start Point

D:0 T:1 F: 0.0 0:00 INCH ABS	
MILL ARC	
RADIUS	
	0.0000
STEP	
	0.0000
	RUN HELP

Enter the incline mill arc radius.

Fig. I.25 Entry Form: Radius

- The contour view shows the position of the tool relative to the milling surface. When the crosshair representing the tool is on the line representing the surface, the tool is in position. The tool crosshair remains fixed in the center of the graph. As the table is moved, the surface line moves.
- Press the **END** soft key to exit the milling operation.



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.



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 – 3 Turning Specific Operations

This section discusses operations specific to turning applications only.

Tool Table

The VUE can store the dimensional offsets for up to 16 tools. When you change a workpiece and establish a new datum, all tools are automatically referenced from the new datum.

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 features. See the following examples for instructions on Tool Offsetting (See Fig. I.26).

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.

Setting Tool Offsets Example 1: Using TOOL/SET

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 T00L hard 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 VUE is in diameter display mode (Ø) if you input 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.

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

TOOL TABLE (X/Z)	
1	1.5200Ø
2	
3	
4	
5	2.4500Ø
6	
7	
8	

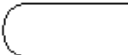


  

Fig. I.26 Tool table in turning

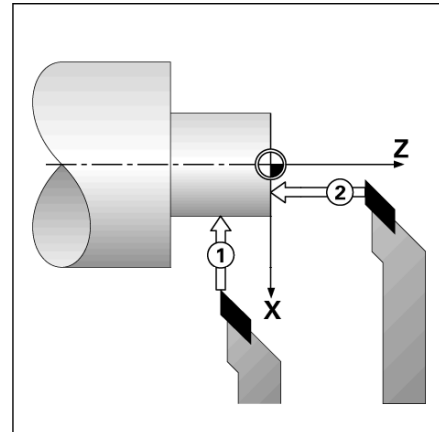


Fig. I.27

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. See Fig Fig. I.28.

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 T00L hard 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. See Fig Fig. I.29

Remember to ensure the VUE 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 T00L hard key. Use the UP/DOWN ARROW keys to cursor through the selection of tools (1-16). Highlight the tool you want. Verify the proper tool has been called, and press either the **USE TOOL** hard key, or the C key to exit.

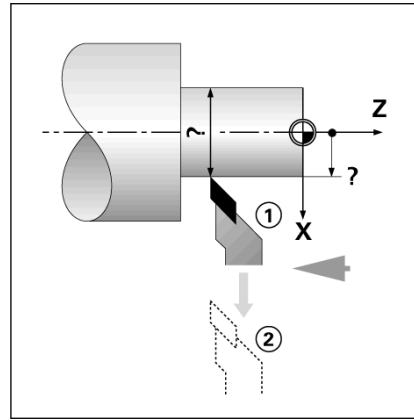


Fig. I.28 Setting Tool Offset

D:0 T:1 F: 0.0 0:00 INCH ABS SET	
TOOL/SET	X 0.0000 Ø
TOOL	Z 0.0000
X 0.0000 Ø	Turn dia. in X then press LOCK AXIS or enter tool position.
Z	
LOCK AXIS	HELP

Fig. I.29 TOOL/SET Form

Datum Setting

See "Datum Setting" on page 8 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 easiest 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. See Fig. I.30.

Preparation:

Call the tool data by selecting the tool which you are using to touch the workpiece. Press the **DATUM** hard 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 VUE is in diameter display mode (\emptyset) if you input a diameter value. Press the **DOWN ARROW** key to advance to the Z axis.

Touch the workpiece surface at point 2. Enter the position of the tool tip (Z= 0) for the Z coordinate of the datum. Press **ENTER**.

Setting Datums using LOCK AXIS Function

The **LOCK AXIS** function is useful for setting a datum when a tool is under load and the diameter of the workpiece is not known. See Fig I.31.

To use the **LOCK AXIS** function:

Press the **DATUM** hard 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**.

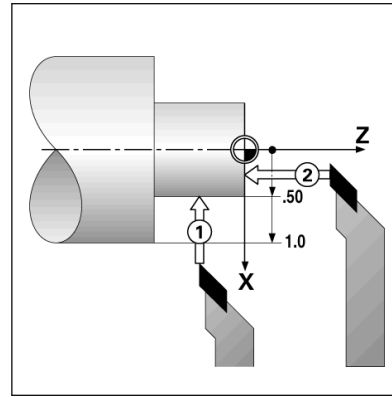


Fig. I.30 Setting a workpiece datum

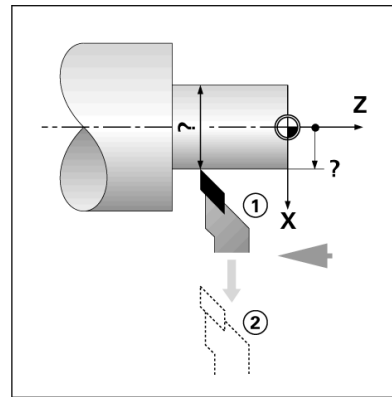


Fig. I.31

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

SET DATUM		X	-2.1842 \emptyset	X
DATUM NUMBER		Z	0.0000	
DATUM		Turn dia. in X then press LOCK AXIS or enter tool position.		
X	\emptyset			
Z		Z		
LOCK AXIS			HELP	

Fig. I.32 Setting Datum using LOCK AXIS

Taper Calculator Hard Key

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

Use the taper calculator to calculate taper angle. See Fig. I.33 and Fig. I.34.

Entry values:

For the taper ratio, calculation requires:

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

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

- Starting diameter
- End diameter
- Length of the taper

Press the CALC hard key.

The soft key selection now changes to include the taper calculator functions.

To calculate the taper angle using two diameters, and length between, press the taper: di/D2/l soft keys. First taper point, diameter 1, either enter a point using the numeric keys and press enter, or touch the tool to one point and press note.

Repeat this for the diameter 2 field.

When using the note key, the taper angle is automatically calculated. When entering data numerically, enter data into the length field, and press enter. The taper angle will appear in the angle field.

To calculate angles using the ratio of the diameter, change to length, and press the **TAPER: RATIO** soft key.

Using the numeric keys, enter data into the ENTRY 1, and ENTRY 2 fields. Press ENTER after each selection. The calculated ratio, and the angle will appear in their respective fields.

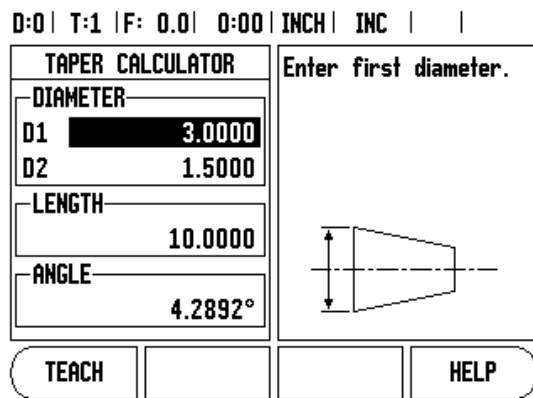


Fig. I.33 Taper Calculator form - Diameter 1

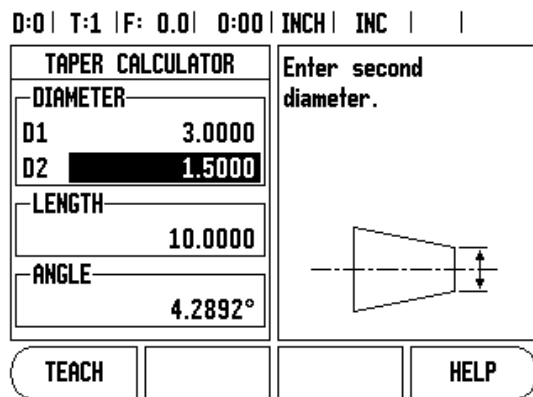


Fig. I.34 Taper Calculator form - Diameter 2

Presets

The functionality has been explained previously in this manual (See "Presets" on page 10). 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. VUE 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. See Fig. I.35.

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.

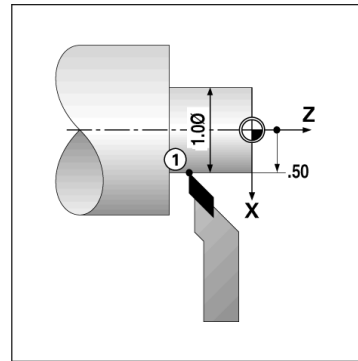


Fig. I.35 Workpiece for radius/diameter display

Vectoring

Vectoring breaks down the movement of the compound axis into the crossfeed or longitudinal axes. See Fig. I.36. 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.

Press the VECTORIZING hard key.

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.

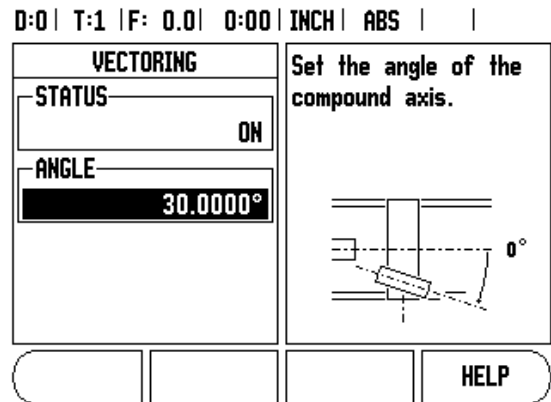


Fig. I.36 Vectoring

II – 1 Installation Setup

Installation Setup Parameters

Installation setup is accessed by pressing the **SETUP** soft key, which brings up **INSTALLATION SETUP** soft key. See Fig. II.1.

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

See "Readout Parameter Access Code" on page iii

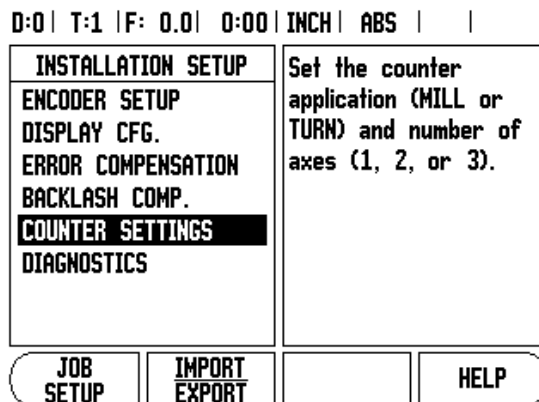


Fig. II.1 Installation screen

Encoder Setup

The ENCODER SETUP is used to set the encoder resolution and type (linear, rotary), count direction, reference mark type. See Fig. II.2.

- ▶ The cursor will default to the **ENCODER SETUP** field upon opening Installation Setup. Press ENTER. This opens a list of possible encoder inputs.
- ▶ Scroll to the encoder you want to change and press ENTER.
- ▶ Cursor will be in the ENCODER TYPE field, select the encoder type by pressing the **LINEAR/ROTARY** soft key.
- ▶ For linear encoders, cursor to the RESOLUTION field and use **COARSER** or **FINER** soft keys to select the encoder's resolution in μm (10, 5, 2, 1, 0.5) or type in the exact resolution. For rotary encoders, enter the number of counts per revolution.
- ▶ In the REFERENCE MARK field, toggling the **REF MARK** soft key selects whether the encoder has no reference signal with **NONE**, single reference mark with the **SINGLE** or with the **P-TRAC** soft key for encoders with the Position-Trac™ feature.
- ▶ In the COUNT DIRECTION field, select the count direction by pressing the **POSITIVE** or **NEGATIVE** soft key. If the encoder's count direction matches the user's count direction, select positive. If the directions do not match, select negative.
- ▶ In the ERROR MONITOR field, select whether the system will monitor and display encoder errors by selecting **ON** or **OFF**. When an error message occurs, press the C key to remove it.

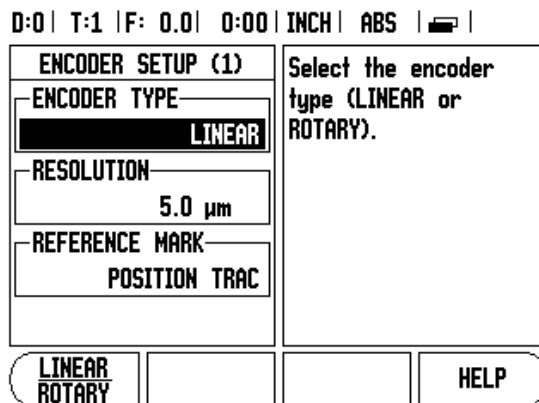


Fig. II.2 Encoder Setup form



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

Display Configuration

The DISPLAY CONFIGURATION form is where the operator determines which axes are displayed and in what order.

- ▶ 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.
- ▶ 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.

Coupling

- ▶ Press the numeric keys 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"). See Fig. II.4.

Z Coupling (turning applications only)

The VUE Turning application provides a quick method for coupling the Z₀ and Z axis position on a 3 axis system. The display can be coupled in either the Z or Z₀ displays. See Fig. II.3.

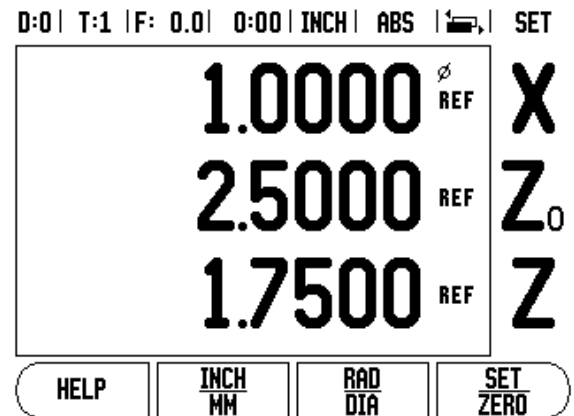


Fig. II.3 Normal Display form

Enabling Z Coupling

To couple the Z₀ and Z axis and have the result displayed on the Z₀ display, press and hold the Z₀ key approximately 2 seconds. The sum of the Z positions will be displayed on the Z₀ display and the Z display will be blanked. See Fig. II.4.

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

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

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

Disabling Z Coupling

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

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, ex. 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 VUE 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.

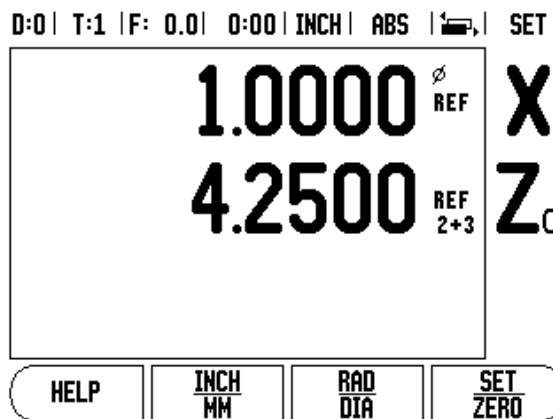


Fig. II.4 Enabling Z Coupling

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. See Fig. II.5 & Fig. II.6.

- ▶ 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.

To calculate the linear error compensation use this formula:

$$\text{Correction factor LEC} = \left(\frac{S - M}{M} \right) \times 10^6 \text{ ppm}$$

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).

$$\text{LEC} = \left(\frac{500 - 499.95}{499.95} \right) \times 10^6 \text{ ppm}$$

LEC = 100 ppm
(rounded to the nearest whole number)

Fig. II.5 Linear error comp., calculation formula

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. VUE supports up to 200 points per axis. The error value between two entered adjacent correction points is calculated with linear interpolation.



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

Starting a Non-linear Error Compensation Table

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

Configuring the Compensation Table

- ▶ Press the **EDIT TABLE** soft key to view the table entries.

D:0 T:1 F: 0.0 0:00 INCH ABS			
ERROR COMPENSATION		Error compensation for this input is OFF.	
INPUT 1	0 PPM	Press TYPE to select linear or non-linear error compensation.	
INPUT 2	OFF		
INPUT 3	OFF		
TYPE [OFF]			HELP

Fig. II.6 Linear error compensation form

- ▶ 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 USB port.

Exporting the Current Compensation Table

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

Importing a New Compensation Table

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

Backlash Compensation

When using a rotary encoder with a lead screw, a change in direction of the table might cause an error in the displayed position due to clearances within the lead screw assembly. This clearance is referred to as backlash. This error can be compensated for by inputting the amount of backlash within the lead screw into the Backlash Compensation feature. See Fig. II.7.

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.

D:0 T:1 F: 0.0 0:00 INCH ABS			
BACKLASH COMP.		Specify the amount of backlash between the encoder and the machine.	
INPUT 1	0.2		
INPUT 2	OFF		
INPUT 3	OFF		
ON OFF			HELP

Fig. II.7 Backlash compensation form

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. See Fig. II.8.

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. The operator will be prompted to press **YES** to set parameters to factory default settings or **NO** to cancel and return to previous menu screen.

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. See "Reference Mark Evaluation" on page 3.

Diagnostics

The DIAGNOSTICS menu provides access for testing the keypad and edge finders. See Fig. II.9.

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.

Display Test

- ▶ To test the display, press the enter key to set the display to solid black, solid white, and back to normal.

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

COUNTER SETTINGS		Set the application to MILL or TURN. Press FACTORY DEFAULTS to reset all parameters to their factory default settings.
APPLICATION	MILL	
NUMBER OF AXES	3	
POSITION RECALL	OFF	
MILL TURN/GP	FACTORY DEFAULTS	HELP

Fig. II.8 COUNTER SETTINGS form

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

	Use the graphic to verify a key's press and release. To exit, press the clear key (C) twice.

Fig. II.9 DIAGNOSTICS form

II – 2 Installation and Electrical Connections

Installation

The DRO is mounted to a tilt/swivel feature: See "Dimensions" on page 32.

Electrical requirements

Voltage 100 - 240 Vac

Power 25 VA max.

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

Degree of protection (EN 60529) IP 40 back panel

IP 54 front panel

Fuse 500 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 (see Fig. II.10).

Preventative maintenance

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

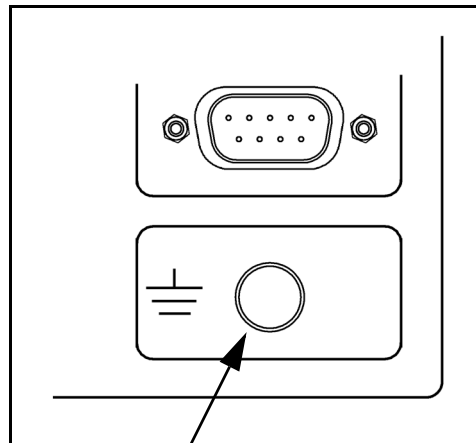
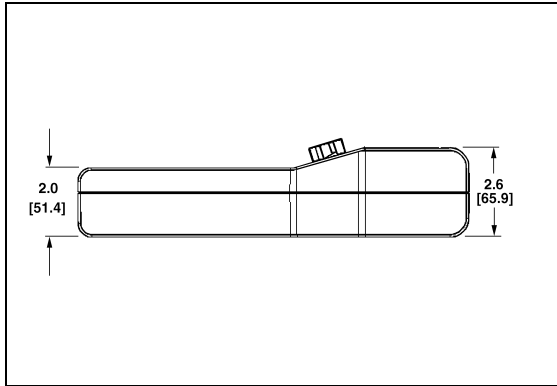


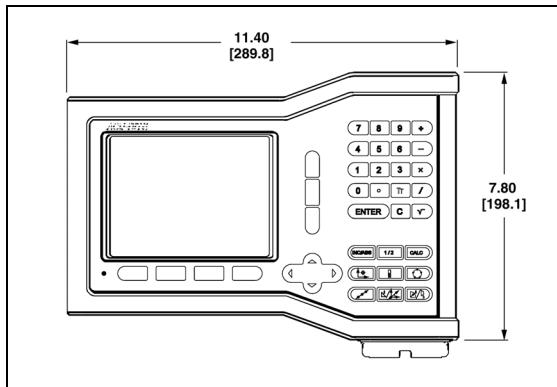
Fig. II.10 The protective conductor terminal (grounding) on the rear panel.

II – 3 Dimensions

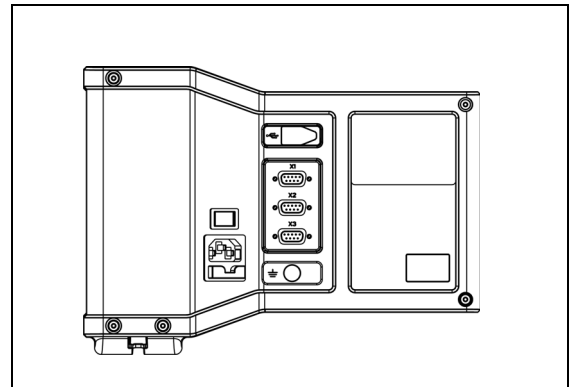
Dimensions in inches/mm



Top view with Dimensions



Front view with Dimensions

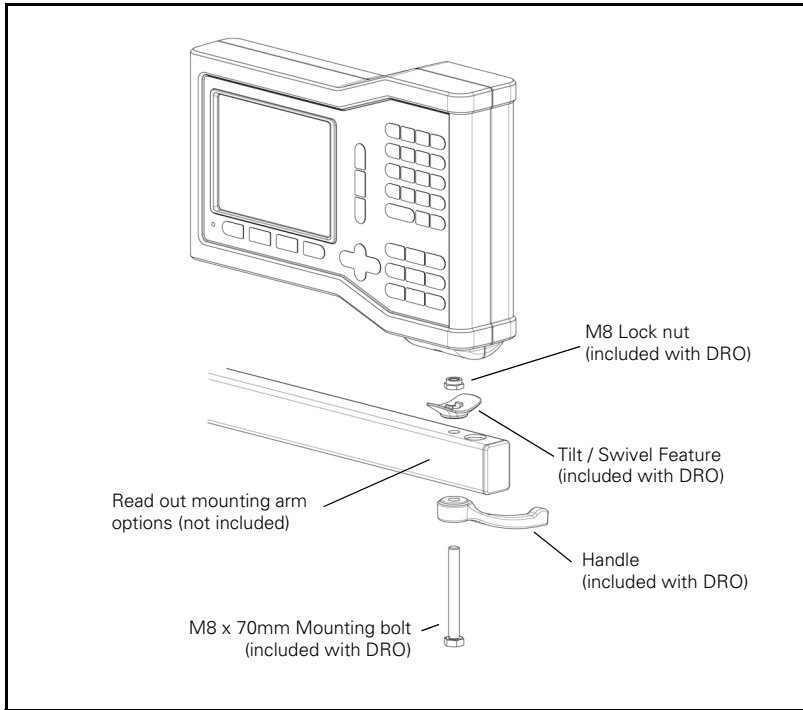


Back view

Accessory ID Number

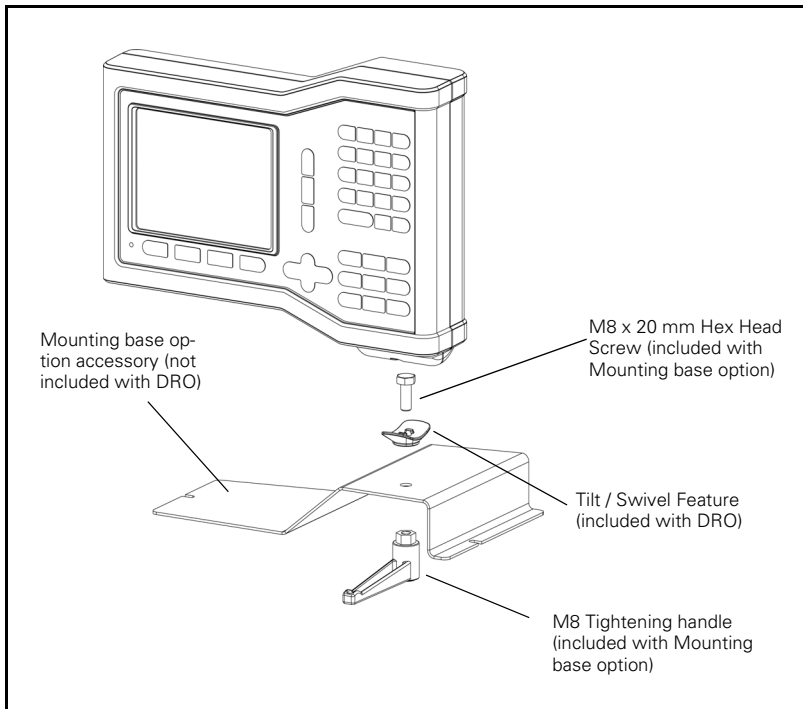
ID Number	Accessory
627052-01	Pkgd, Mounting Base

DRO mounting with arm (reference information)



The DRO mounting base incorporates a slot feature that prevents the lock nut from turning. Pre-attach the mounting bolt and slip it into the base. Secure the DRO by tightening the handle.

DRO mounting with base



Numerics

1/2 Hard Key 13

1/2 soft key 13

A

Absolute 3

Absolute Distance Preset 10

Actual Value/Distance-To-Go soft key 3

Axis Labels 1

B

Backlash Compensation 29

C

CENTER LINE soft key 9

Circle and Linear Pattern 13

CIRCLE CENTER soft key 9

Console adjustment 5

Counter Settings 27, 30

D

Datum setting (Turning) 22

Datum soft key (Milling) 8

Datum Soft Key (Turning) 22

Diagnostics 30

Diameter Axes (Milling) 5

Dimensions 32

Disable Ref soft key 4

Display Area 1

Display configuration 26

E

EDGE soft key 9

Electrical Requirements 31

Enable Ref soft key 3

Enable/Disable Ref Function 4

Enable/Disable soft key 4

Encoder Setup 25

Environmental specs. 31

Error Compensation 27

G

General Operation's Soft Key Function Detailed 6

Grounding 31

I

Import/Export (setting) 6

IMPORT/EXPORT soft key 6

Incline & Arc Milling 15

Incremental 3

Incremental Distance Preset 12

Installation Setup Parameters 25

INSTALLATION SETUP soft key 25

J

Job Clock 5

Job Setup Parameters 4

L

Language (setting) 6

LANGUAGE soft key 6

Layout of Screen 1

Linear Error Compensation 28

Linear Pattern 15, 16, 18

M

Milling Specific Operations and Soft Key Functions Detailed 7

Mirror 4

N

Near Zero Warning 5

No Ref soft key 3

Non-Linear Error Compensation 28

O

Operating Modes 3

P

Patterns (Milling) 13

Position Recall 30

Preset 10

Preset function 10

Preset Soft Key (Turning) 24

Preventative Maintenance 31

R

Radius/Diameter display 23, 24

Ratio soft key 23

Ref Symbols 1

Reference Mark Evaluation 3

Reference marks

 crossing over 3

 not crossing over 3

S

Scale factor 4

Set Zero soft key 6

SET/ZERO soft key 6

Soft key Labels 1

Status Bar 1

Status Bar (setting) 5

T

Taper Calculator 23

TOOL hard key 20

Tool Hard Key 7

Tool setting, turning 20

Tool soft key 7

Tool soft key (Turning) 20

U

Units of measurement, setting 4

V

Vectoring 24

Z

Z Coupling 26

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

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

