



HEIDENHAIN

User's Manual

ND 522/523



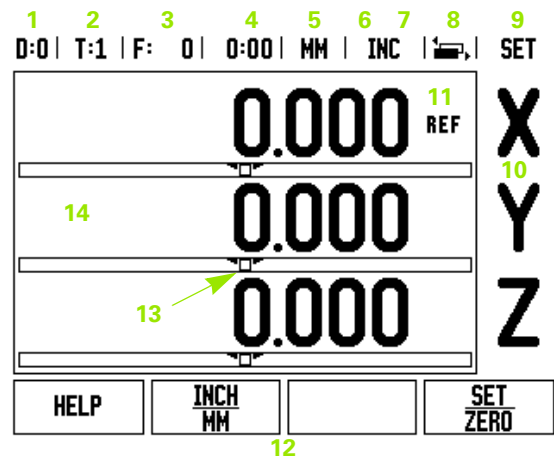
English (en)
7/2014



ND 522/523 Screen

View of the ND 522/523 screen defining typical information provided.

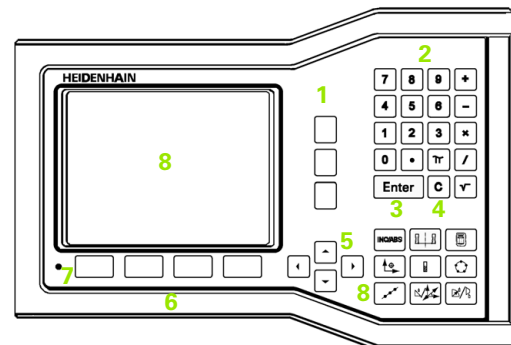
- 1 Datum
- 2 Tools
- 3 Feed Rate
- 4 Stopwatch
- 5 Unit of Measure
- 6 Actual Value
- 7 Distance-To-Go
- 8 Page Indicator
- 9 Set/Zero
- 10 Axis Label
- 11 Ref Symbol
- 12 Soft Key Functions
- 13 Graphic Positioning Aid
- 14 Display Area



ND 522/523 Front Panel

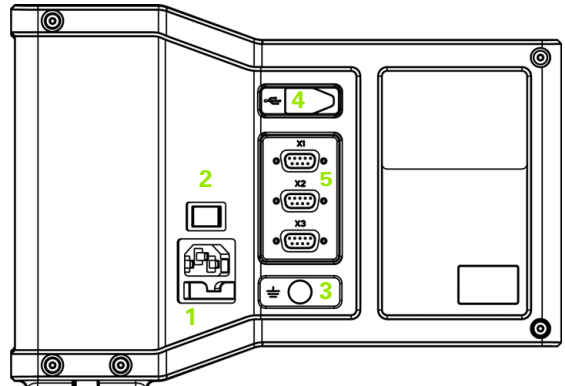
View of the ND 522/523 front panel defining keys, and features.

- 1 Axis keys (3) - X, Y, & Z
- 2 Numeric input keys
- 3 **Enter** key, confirm entry, and select entry field
- 4 CLEAR key
- 5 ARROW keys
- 6 SOFT KEYS that functions vary depending on the current screen shown
- 7 Power indicator light
- 8 Display screen
- 9 Hard Function Keys



ND 522/523 Back Panel

- 1 Main power input
- 2 Power switch
- 3 Grounding Edge Finder
- 4 KT 130 Edge Finder
- 5 Ground (Protective Earthing)
- 6 Serial Port
- 7 Axis ports





Introduction

Software Version

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



This User's Manual covers the functions of the ND 522/523 for both **milling**, and **turning** applications. Operational information is arranged in three sections: General Operations, Mill Specific Operations, and Turn Specific Operations.

ND 522/523

DRO axis availability.



The ND 522/523 DRO is available in **three axis** form only. The 3 axis ND 522/523 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 ND 522/523.



Warning Refer to accompanying documents

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



Caution - Risk of electric shock, or "in harms way"

e.g. when opening a housing.



Different

e.g. from machine to machine.

Manual Ref.

e.g. Refer to a different manual.

ND 522/523 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



I Operating Instructions 11

I - 1 Fundamentals of Positioning	12
Datums	12
Actual Position, Nominal Position, and Distance-To-Go	12
Absolute Workpiece Positions	13
Incremental Workpiece Positions	13
Zero Angle Reference Axis	14
Position Encoders	14
Encoder Reference Marks	15
I - 2 General Operations for ND 522/523	16
Screen Layout	16
General Operation Hard Key Function Overview	17
General Navigation	18
Operating Modes	18
Graphic Positioning Aid	19
Help Screen	19
Data Input Forms	20
Instruction Box messages	20
Error Messages	20
Power Up	21
Reference Mark Evaluation	21
Working without reference mark evaluation	21
ENABLE/DISABLE REF function	22
Setup	23
Job Setup Parameters	23
Units	23
Scale Factor	24
Mirror	24
Diameter Axes	24
Graphic Positioning Aid	25
Status Bar Settings	25
Stopwatch	25
Console Adjustment	26
Language	26
Import/Export	26
DRO Operating Screen Soft Key Function Overview	27
General Operation Key Functions Detailed	28
Set Zero Soft Key	28
1/2 Hard key	29
Calc Hard key	30

I - 3 Milling Specific Operations	31
Key Functions Detailed	31
Tool Hard Key	31
Import/Export	31
Tool Radius Compensation feature	32
Tool Length	32
Sign for the length difference ΔL	32
Entering tool data	32
Tool Table Usage	33
Calling the Tool Table	35
Datum Hard key	35
Probing with a Tool	37
Presetting	39
Absolute Distance Preset	39
Incremental Distance Preset	43
RPM Calculator	45
Circle, and linear Patterns (Milling)	46
Functions for milling patterns	46
Circle Pattern	46
Linear Pattern	49
2nd step: Drill	51
Incline & Arc Milling	52
Incline Milling	52
Arc Milling	55
I - 4 Turning Specific Operations	58
Keys Functions Detailed	58
Turning Specific Display Icons	58
Tool Hard key	58
To access the Tool Table menu:	58
Tool Table Usage	59
Setting tool offsets using TOOL/SET	59
Setting Tool Offset using NOTE/SET Function	60
Datum Hard key	61
Example: Setting a workpiece datum.	61
Preparation:	61
Setting Datums using NOTE/SET Function	63
Taper Calculator Hard Key	64
Presetting	65
RX (Radius/Diameter) Soft Key	65
Vectoring Hard Key	66
Z Coupling (turning applications only)	67
Enabling Z Coupling	67
Disabling Z Coupling	67

II Technical Information 69

II - 1 Installation, and Electrical Connection	70
Items Supplied	70
Accessories	70
ND 522/523 Display Unit	70
Mounting Location	70
Installation	70
Electrical connection	70
Electrical requirements	71
Environmental	71
Wiring the power connector	71
Protective earthing (grounding)	71
Preventative maintenance	71
Connecting the Encoders	72
II - 2 Installation Setup	73
Installation Setup Parameters	73
Encoder Setup	73
Display Configuration	74
Coupling	74
Error Compensation	74
Linear Error Compensation	75
Non-Linear Error Compensation	75
Configuring the Compensation Table	76
Reading the Graph	76
Viewing the Compensation Table	76
Exporting the Current Compensation Table	76
Importing a New Compensation Table	76
Backlash Compensation	77
Counter Settings	78
Diagnostics	78

II - 3 Encoder Parameters	79
Example settings for HEIDENHAIN linear encoders	79
Example settings for HEIDENHAIN Rotary encoders	79
Example settings for HEIDENHAIN Angle encoders	79
II - 4 Data Interface	80
USB Port (type "B")	81
External Operations via USB port	81
II - 5 Measured Value Output	84
Examples of character output at the data interface	84
II - 6 Specifications for Milling	86
II - 7 Specifications for Turning	87
II - 8 Dimensions	88
II - 9 Accessories	89
Accessory ID Numbers	89
ND 522/523 Handle	
ID 618025-01	89



Operating Instructions



I - 1 Fundamentals of Positioning

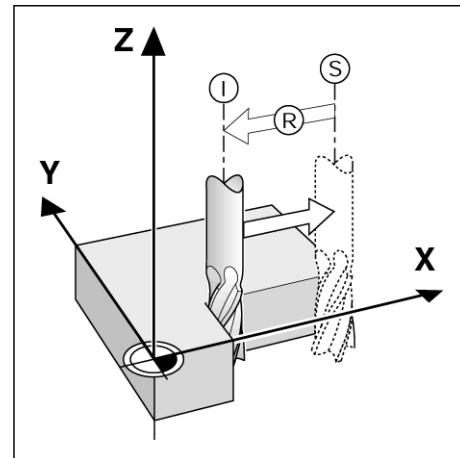
Datums

The workpiece drawing identifies a certain point on the workpiece (usually 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, and 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 **S** is called the **nominal position**. The distance from the nominal position to the actual position **R**, is called the **distance-to-go** (Incremental).



Absolute Workpiece Positions

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

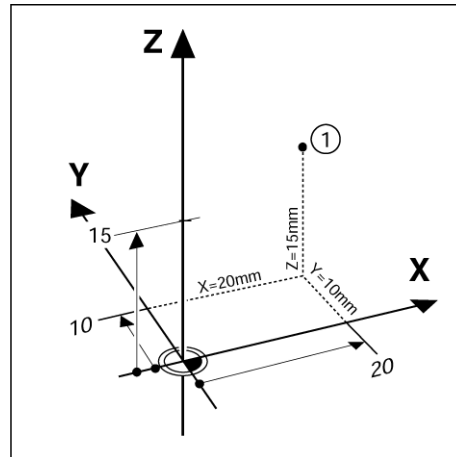
Example: Absolute coordinates of position **1**:

X = 20 mm

Y = 10 mm

Z = 15 mm

When drilling, or milling a workpiece according to a workpiece drawing with **absolute coordinates**, the tool is moving 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

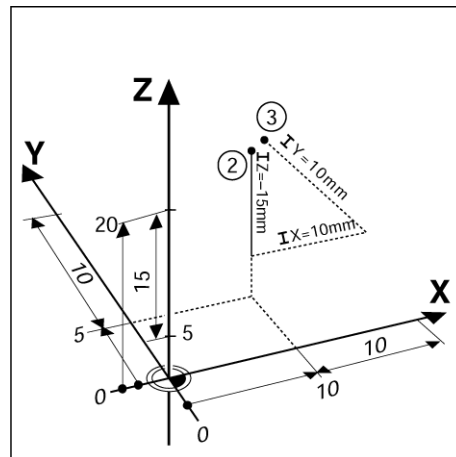
Incremental coordinates of position **3**:

IX = 10 mm

IY = 10 mm

IZ = -15 mm

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



Zero Angle Reference Axis

The Zero Angle Reference Axis is the 0 degree 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.

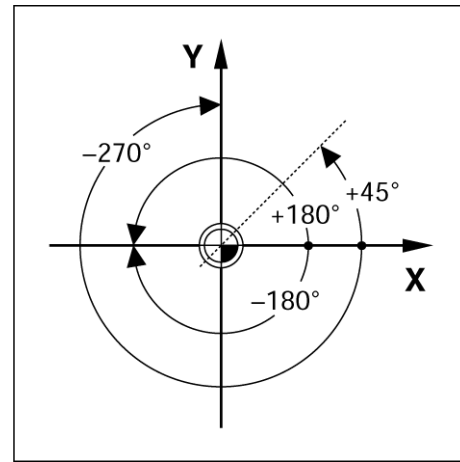
For angular positions, the following reference axes are defined:

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

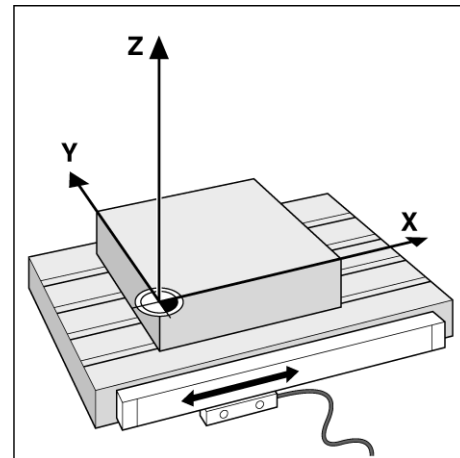
Angle	Corresponds to the...
+ 45°	... bisecting line between +X, and +Y
+/- 180°	... negative X axis
- 270°	... positive Y axis



Position Encoders

The **position feedback** encoders convert the movement of the machine axes into electrical signals. The ND 522/523 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, this relationship can be re-established with the aid of the reference marks on the position encoders, and the ND 522/523's reference mark evaluation feature (**REF**).

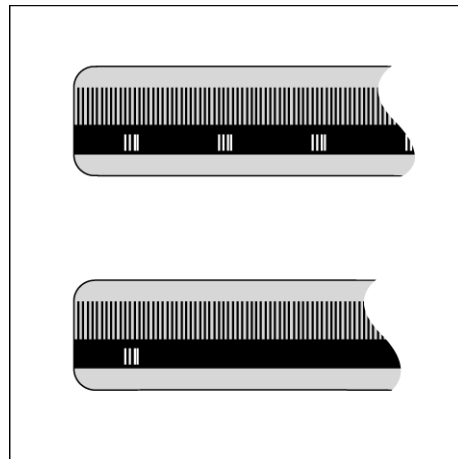


Encoder Reference Marks

Encoders normally contain one, or more reference marks which the ND 522/523's 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.

Encoders with **distance-coded reference marks** have marks separated by a specific encryption pattern that allows the ND 522/523 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 a very short distance, any where along the encoder, to re-establish the datums when the ND 522/523 is turned back on.

Encoders with **fixed reference marks** have one, or more marks on fixed intervals. 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 ND 522/523

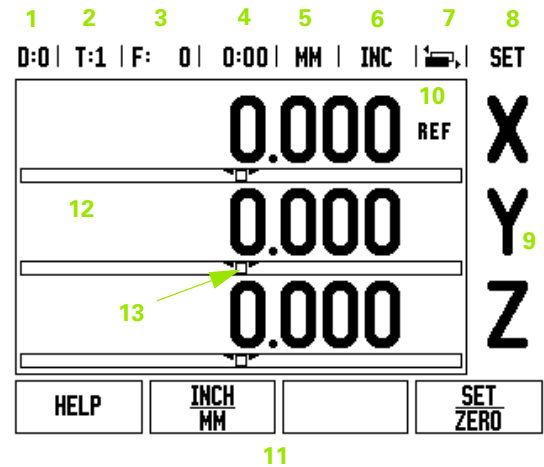
Screen Layout

Status Bar Symbols

- 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 ND 522/523 readout provides application-specific features that allows the most productivity be obtained from 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 Operation Hard Key Function Overview

The following is a list of Hard keys, and a description of their function which are located on the front panel of the readout.

Hard Key Page 1	Hard Key function	Hard key Symbol
INCREMENTAL /ABSOLUTE	Switches display between Distance-To-Go (Incremental) Actual Value (Absolute). (page 18)	
1/2 (Mill Function Only)	Used to divide the current position by two. (page 29)	
CALC	Opens the Calculator functions. (page 30)	
DATUM	Opens the Datum form to set the datum for each axis. (page 35)	
TOOL	Opens the Tool Table. (page 31 for Milling. page 58 for Turning)	
CIRCLE PATTERN	Opens the Circle Pattern form. This calculates the hole positions (page 46) for Milling	
LINEAR PATTERN	Opens the Linear Pattern form. This calculates the hole positions (page 49) for Milling	
INCLINE MILLING, or VECTORING	Opens the Incline milling form (page 52) for Milling, or the Vectoring form (page 66) for Turning	
ARC MILLING, or TAPER CALC	Opens the Arc milling forms (page 55) for Milling, or the Taper Calc form (page 64) for Turning	



General Navigation

- Use keypad 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 2 pages of selectable soft key functions. These are accessed using the LEFT/RIGHT ARROW keys.
- The LEFT/RIGHT ARROW keys move through pages 1, and 2 of the soft key selectable functions. The current page will be highlighted in the Status bar at the top of screen.
- Use the UP/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 the menu.

Operating Modes

The ND 522/523 has two operating modes: **Distance-To-Go (incremental)**, and **Actual Value** (absolute). The Distance-To-Go feature (which will be referred to as incremental in this manual) enables approach to nominal positions by traversing to display value zero. When working within the incremental mode either enter nominal coordinates as either incremental, or absolute dimensions. The Actual Value feature (which will be referred to as absolute in this manual) 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.

While in the Absolute Mode, if the ND 522/523 is configured for Milling applications, only the tool length offsets are active. Both the radius, and length offsets are used in the 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 ND 522/523 is configured for a lathe, all tool offsets are used in both the Incremental, and Absolute modes.

Press the INCREMENTAL / ABSOLUTE hard key to toggle between these two modes. To view soft key functions in either Incremental, or Absolute 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. See "Enabling Z Coupling" on page 67.

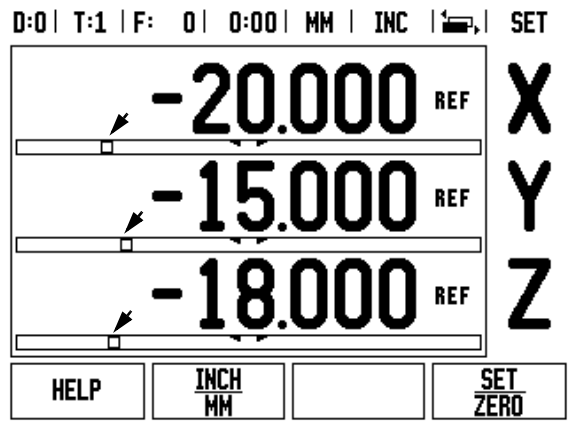


Graphic Positioning Aid

When traversing to display value zero (in the incremental mode), ND 522/523 displays a graphic positioning aid.

ND 522/523 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 to be reached.

A small square symbolizes the axis slide. An arrow indicating the direction appears in the square while the axis is moving. This indicates if 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. For setting up the graphic positioning aid. See "Graphic Positioning Aid" on page 25 under Job Setup.



Help Screen

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

To **call** the operating instructions:

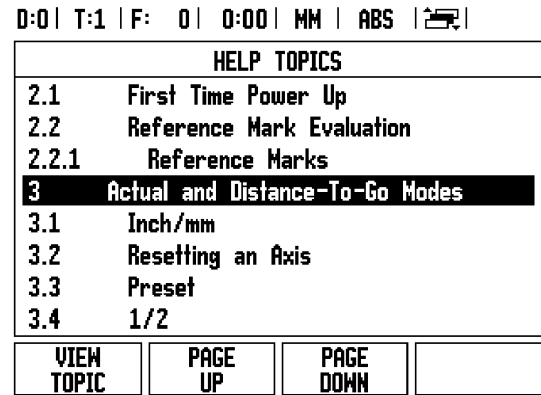
- ▶ Press the HELP soft key.
- ▶ Information relevant to the current operation will be displayed.
- ▶ Use the UP/DOWN ARROW keys if the explanation is spread over more than one screen page.

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 an item.

To **leave** 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.

Confirm changes by pressing the **Enter** key, and for them to become effective. If changes are not to be saved, press the C key to return to the previous screen without saving changes.

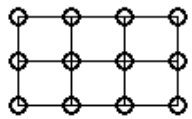
Instruction Box messages

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

D:0 | T:1 | F: 0 | 0:00 | MM | INC | 

LINEAR PATTERN	
TYPE	ARRAY
FIRST HOLE	
X	20.000
Y	15.000
HOLES PER ROW	4

Select the pattern type (ARRAY or FRAME).



ARRAY FRAME			HELP
----------------	--	--	------

Error Messages

If an error occurs when working with ND 522/523, the message will appear on the display, and provide an explanation of what caused the error.

To **clear** the error message:

- ▶ Press the C key.

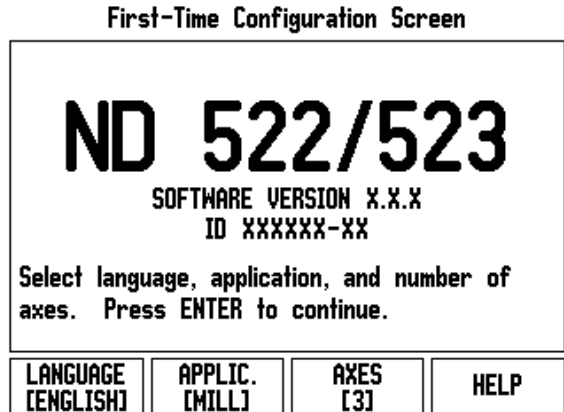
Power Up



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 the 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, the application can be changed later in "Installation Setup" under "Counter Settings".



The ND 522/523 is now ready for operation, and is in the operating mode Absolute. Each active axis will have a flashing "REF" sign next to it. At this point the reference mark evaluation should be completed.

Reference Mark Evaluation

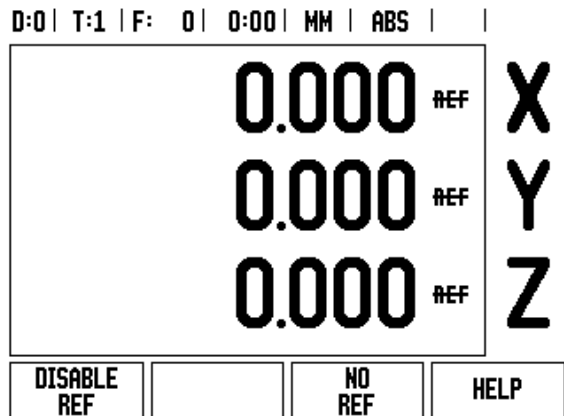
The ND 522/523's reference mark evaluation feature automatically re-establishes the relationship between axis slide positions, and display values that were last defined by setting the datum.

If the axis encoder has reference marks, the "REF" indicator will flash. After crossing over the reference marks, the indicator will stop flashing, and change to non-flashing REF.

Working without reference mark evaluation

The ND 522/523 can be used without crossing over the reference marks. Press the NO REF soft key to exit the reference mark evaluation routine, and continue.

Reference marks can be crossed over at a later time, if it becomes necessary to define datums that can be re-established after a power interruption. Press the ENABLE REF soft key to activate the reference mark evaluation routine.



If an encoder is setup without reference marks, then the REF indicator will not be displayed, and datums will be lost once power is turned off.



ENABLE/DISABLE REF function

The toggling ENABLE/DISABLE soft key, that is present during the Reference Mark Evaluation routine, allows the operator to select a specific Reference Mark on an encoder. This is important when using encoders with Fixed Reference Marks. When the DISABLE REF soft key is pressed, the evaluation 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 evaluation routine once again becomes active, and the next crossed reference mark will be selected.

Once reference marks for all desired axes are established, press NO REF soft key to cancel out of routine. Only the axes that are needed require crossing over the reference marks. If all reference marks have been found the ND 522/523 will return to the DRO display screen automatically.



If the reference marks are **not** crossed over, the ND 522/523 does not store the datum points. This means that it is not possible to re-establish the relationship between axis slide positions, and display values after a power interruption (switch-off).



For everyday power up; turn on power, and press any key.

Cross over the reference marks (in any order).

- ALTERNATIVE METHOD -

**DISABLE
REF**

Press DISABLE REF soft key, and cross over reference marks.

**ENABLE
REF**

Move encoder to desired fixed reference mark. Press ENABLE REF soft key, and cross over reference mark.

- ALTERNATIVE METHOD -

**NO
REF**

Do **not** cross over the reference marks, and press the NO REF soft key. Note: In this case the relationship between axis slide position, and display value will be lost after a power interruption.



Setup

ND 522/523 offers two categories for setting up operating parameters. These categories are: Job Setup, and Installation Setup. The Job Setup parameters are used to accommodate specific machining requirements for each job. Installation Setup is used to establish encoder, and display parameters.

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

■ INSTALLATION SETUP

Press to begin accessing the Installation Setup parameters. See "Installation Setup Parameters" on page 73.

■ IMPORT/EXPORT

Press to begin importing, or exporting operating parameters. See "Import/Export" on page 26.

■ HELP

Will open on-line help.

D:0 T:1 F: 0 0:00 MM ABS	
JOB SETUP	
UNITS	Set the working units of measure for linear and angular dimensions.
SCALE FACTOR	
DIAMETER AXES	
GRAPHIC POS. AID	
STATUS BAR SETTINGS	
STOPWATCH	
CONSOLE ADJUSTMENT	
LANGUAGE	
INSTALL. SETUP	IMPORT EXPORT
	HELP

Job Setup Parameters

To view, and change Job Setup parameters use the UP/DOWN ARROW keys to highlight the parameters of interest, and press the **Enter** key.

Units

The UNITS form is used to specify the preferred display units, and format. The system powers up with these settings in effect.

- ▶ Inch/MM - Measurement values are displayed, and entered in the units selected in the LINEAR field. Choose between inch, or millimeter by pressing the INCH/MM soft key. Also the unit of measure can be selected by pressing the INCH/MM soft key in either Incremental mode, or Absolute.
- ▶ Decimal Degrees, Radians, or Degrees/Minutes/Seconds (DMS) - The ANGULAR field affects how angles are displayed, and entered into forms. Choose between DECIMAL DEGREES, RADIANS, or DMS using the soft key.



Scale Factor

The scale factor may be used to scale the part up, or down. All encoder movements are multiplied by the scale factor. A scale factor of 1.0 creates a part with the exact size as dimensioned on the print.

- ▶ 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. A part can be both mirrored, and scaled at the same time.

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. For turning applications see page 65 for the Radius/Diameter feature.

- ▶ Cursor to DIAMETER AXES, and press **Enter**.
- ▶ The cursor will be in the X field. Depending on the parameter used for that axis press the ON/OFF soft key to turn feature on, or off.
- ▶ Press **Enter**.

D:0 | T:1 | F: 0 | 0:00 | MM | ABS | |

SCALE FACTOR	
SCALE FACTOR	
X	OFF
Y	OFF
Z	OFF

Set a scale factor to enlarge, shrink, or mirror a part.

When scale factor is ON, the ∇ indicator appears on the axis display.

ON OFF			HELP
-----------	--	--	------

D:0 | T:1 | F: 0 | 0:00 | MM | ABS | |

DIAMETER AXES	
DIAMETER AXES	
X	ON
Y	OFF
Z	OFF

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

ON OFF			HELP
-----------	--	--	------



Graphic Positioning Aid

The GRAPHIC POSITIONING AID form is used to configure the bar graph that is shown below the axes' display in Incremental 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, stop watch, and page indicator.

- ▶ Press the ON/OFF soft key for each setting to be displayed.

Stopwatch

The stopwatch shows the hours (h), minutes (m), seconds (s). It operates like a stop watch showing elapsed time. (The watch 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 watch if it is running.

D:0 | T:1 | F: 0 | 0:00 | MM | ABS | |

GRAPHIC POS. AID		Set the range used on the graphic positioning aid bar graph. The current position box will begin moving when the position is within range.	
RANGE			
X	5.000		
Y	5.000		
Z	5.000		
ON OFF			HELP



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.

Language

The ND 522/523 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 the selection.

Import/Export

Operating parameter information can be imported, or exported over the USB port (See "USB Port (type "B")" on page 81).

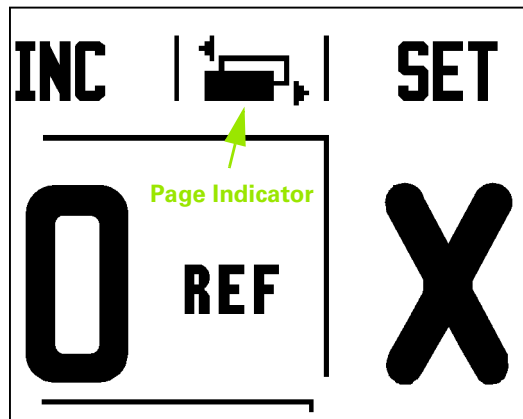
- ▶ Press the IMPORT/EXPORT soft key in the Setup screen.
- ▶ Press IMPORT to download operating parameters from a PC.
- ▶ Press EXPORT to upload the current operating parameters to a PC.
- ▶ To exit, press the C key.



DRO Operating Screen Soft Key Function Overview

There are two pages of soft key functions in the operating screen to select from. 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 currently being viewed.

Soft Key Page 1	Soft Key function	Soft key Symbol
HELP	Opens on-screen help instructions. (page 19)	HELP
INCH/MM	Toggles between inch, and millimeter units. (page 23)	INCH MM
RADIUS/ DIAMETER	Toggles between radius, and diameter displays This function is for Turning applications only. (page 65)	R _x
SET/ZERO	Toggles between Set Zero functions. Used with individual axis keys. (page 28)	SET ZERO
Soft Key Page 2	Soft Key function	Soft key Symbol
SETUP	Opens the Job Setup menu, and provides access to the installation Setup soft key. (page 23)	SETUP
ENABLE REF	Press when ready to identify a reference mark. (page 21)	ENABLE REF



General Operation Key Functions Detailed

This section details the soft key functions that are the same, whether the ND 522/523 is configured for Mill, or Turn applications.

Set Zero Soft Key

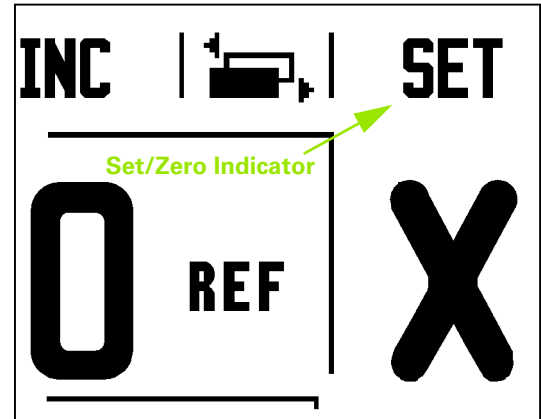
The SET/ZERO soft key is a key that determines the effect of pressing an Axis key. This key is a toggle key, switching the functionality between Set, and Zero. The current state is indicated in the Status Bar.

When the state is Set, and the ND 522/523 is in Absolute mode, selecting an Axis key opens the DATUM form for the selected axis. If the ND 522/523 is in Incremental mode, a PRESET form opens.

When the state is zero, and the ND 522/523 is in Absolute mode, selecting an Axis key sets the datum for that axis to zero at the current position. If it is in Incremental mode, the current incremental value is set to zero.



If the ND 522/523 is in Absolute 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.



1/2 Hard key

The 1/2 hard key is used to find the half-way (or midpoint) between two locations along a selected axis of a workpiece. This can be performed in either Incremental, or Absolute mode.



This feature will change datum locations when in Absolute mode.

Example: Finding the midpoint along a selected axis

X dimension: X = 100 mm

Midpoint: 50 mm

MOVE TO 1ST POINT



Move tool to first point.
SET ZERO soft key must be set to Zero.

ZERO AXIS, AND MOVE TO 2ND POINT



Select the X axis key and

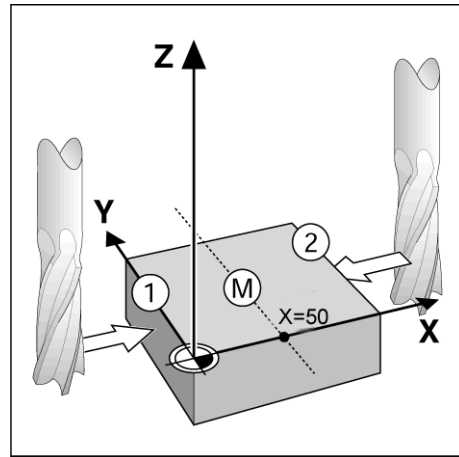


move to second point.

PRESS 1/2, AND MOVE TO ZERO



Press 1/2 hard key, then press the X axis key, and move until zero is reached. This is the midpoint location.



Calc Hard key

ND 522/523's calculator is capable of handling everything from simple arithmetic to complex trigonometry, and RPM calculations.

Press the CALC hard key to access the STANDARD/TRIG, and RPM soft keys.

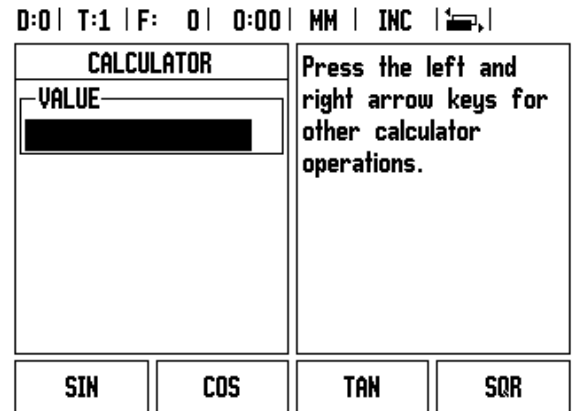
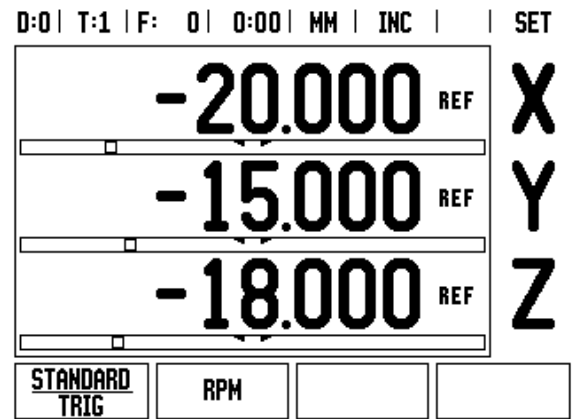


When more than one calculation is entered into a numeric field, the calculator will perform multiplication, and division before it performs addition, and subtraction. If this were to be entered: $3 + 1 \div 8$, ND 522/523 will divide one by eight, then add three for an answer of 3.125.

Trig functions contain all trig operators as well as, square, and square root. When 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, or radians.




I - 3 Milling Specific Operations

This section discusses operations, and soft key functions specific to milling applications only. Soft key functions that are the same, whether the ND 522/523 is configured for Mill, or Turn applications, are detailed starting on page 17.

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. The ND 522/523 can store up to 16 tools within the tool table.

D:0 | T:1 | F: 0 | 0:00 | MM | INC | 


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

TOOL AXIS [Z]	CLEAR TOOL	USE TOOL	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.

D:0 | T:1 | F: 0 | 0:00 | MM | ABS | 

TOOL TABLE (X/Z)	
1	19.082 \emptyset
2	
3	
4	
5	
6	
7	
8	

IMPORT	EXPORT		HELP
--------	--------	--	------



Tool Radius Compensation feature

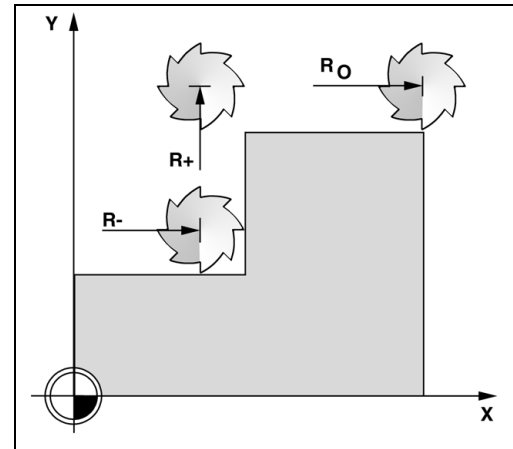
ND 522/523 has a tool radius compensation feature. This allows workpiece dimensions to be entered directly from the drawing. The displayed Incremental is then automatically lengthened (R+), or shortened (R-) by the value of the tool radius. (For more information see page 39).

Tool Length

The length offset may be entered as a known value, or the ND 522/523 may determine the offset automatically. More information is available in the following Tool Table Usage example regarding the TEACH LENGTH soft key.

The following soft keys are 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 effect. 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 the LENGTH field.	TEACH LENGTH
This will open the TOOL TYPES form for selection. Only available in TYPE field.	TOOL TYPES



Sign for the length difference ΔL

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

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

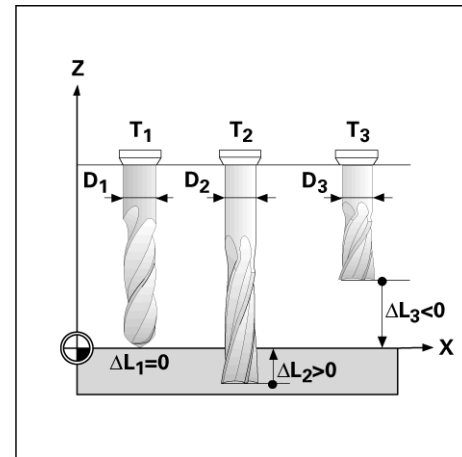


The tool length is the difference in length ΔL between the tool, and the reference tool. The reference tool is indicated by T1.

The length difference is indicated with the “ Δ ” symbol.

Entering tool data

- ▶ Choose tool hard 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.
- ▶ Enter the tool type, and press ENTER to return to the tool table. Press C to exit.



Tool Table Usage

Example: Setting a workpiece datum without using the probing function.

Tool diameter 2.00

Tool length: 20.000

Tool unit: mm

Tool type: flat end mill



It is also possible to have the ND 522/523 determine the length of an offset. See alternative example below.



Press the TOOL hard key.
The cursor will default to the TOOL TABLE form.

TOOL TABLE



Cursor to the tool to be defined, or enter the tool number. Press **Enter**.

TOOL DIAMETER



Enter the tool diameter (2.0mm).



Cursor down to the LENGTH field using the DOWN ARROW key.

TOOL LENGTH



Enter the tool length (20.0mm) .



Cursor down to the UNITS field using the DOWN ARROW key.

D:0 T:1 F: 0 0:00 MM INC			
TOOL		X	0.000
DIAMETER		Y	0.000
2.000		Z	0.000
LENGTH		Enter the tool diameter.	
20.000		X	
UNITS		Y	
MM		Z	
			HELP



- ALTERNATIVE METHOD -

It is also possible to have ND 522/523 determine an offset. This method involves touching the tip of each tool to a common reference surface. This allows ND 522/523 to determine the difference between the length of each tool.

Move the tool until its tip is touching the reference surface.

**TEACH
LENGTH**

Press the TEACH LENGTH soft key. ND 522/523 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.



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, it will not be possible 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 -

**INCH
MM**

Enter the TOOL UNIT (inch/mm).



Cursor down to the TOOL TYPE field.

TOOL TYPE

**TOOL
TYPES**

Press TOOL TYPES soft key. Select from list of tools, and press **Enter**

D:0 | T:1 | F: 0 | 0:00 | MM | INC |

TOOL TYPES			
BALL END MILL	Select the desired tool type then press ENTER.		
BORING HEAD			
BROACH			
CARBIDE MILL			
COUNTER-BORE			
COUNTER-SINK			
DRILL			
ENGRAVING TIP			
			HELP

Calling the Tool Table

Before machining starts, select the tool to be used from the tool table. ND 522/523 then takes into account the stored tool data when working with tool compensation.

Tool call



Press the TOOL hard key.

TOOL NUMBER



Use the UP/DOWN ARROW keys to cursor through the selection of tools (1-16). Highlight the tool to be used.



Verify the proper tool has been called, and press the C key to exit.

Datum Hard key

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

The easiest way to set datum points is to use the ND 522/523's probing function when probing the workpiece with an edge of a tool.

Datum points can also be set in the by touching the edges of the workpiece, one after the other with a tool, and manually entering the tool positions as datum points (see examples following this page).

The datum table can hold up to 10 datum points. In most cases this will eliminate from having to calculate the axis travel when working with complicated workpiece drawings containing several datums.



Example: Setting a workpiece datum without using the probing function.

Tool diameter: $D = 3$ mm

Axis sequence in this example: X - Y - Z

Preparation: Set the active tool to the tool that will be used to set the datum



Press the DATUM hard key.

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 edge **1**.

DATUM SETTING X



Enter the position of the tool center ($X = -1.5$ mm) and



press the DOWN ARROW key to advance to the Y-axis.



Touch the workpiece at edge **2**.

DATUM SETTING Y



Enter the position of the tool center ($Y = -1.5$ mm).



Press the DOWN ARROW key.



Touch the workpiece surface.

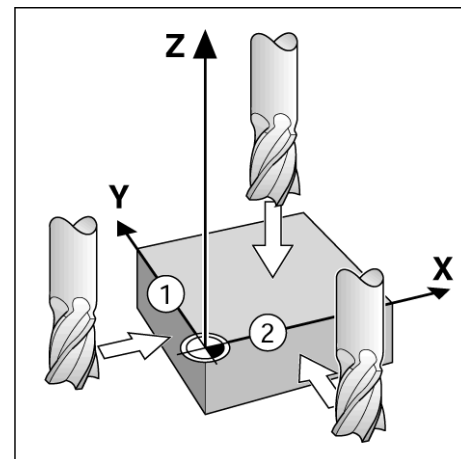
DATUM SETTING Z = + 0



Enter the position of the tool tip ($Z = 0$ mm) for the Z-coordinate of the datum. Press **Enter**.

D:1 T:1 F: 0 0:00 MM ABS	SET
SET DATUM	X 0.000
DATUM NUMBER 1	Y 0.000
DATUM	Z 0.000
X -1.500	Enter the new actual position of the tool or press PROBE.
Y -1.500	
Z 0	
PROBE	HELP

X
Y
Z



Probing with a Tool

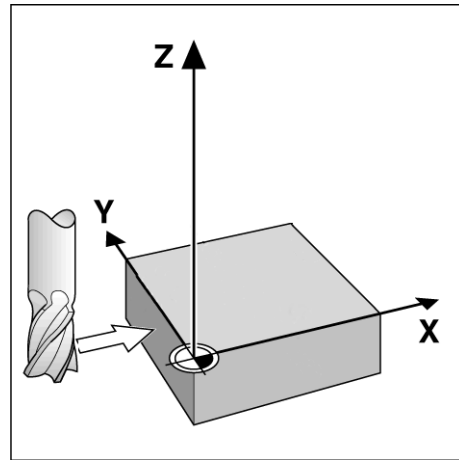
Using a tool to set datum points using the ND 522/523's probing functions.

Preparation: Set the active tool to the tool that will be used to set the datum.

Example: Probe workpiece edge, and set edge as datum

Datum axis: X = 0 mm

Tool diameter D = 3 mm



Press DATUM.



Press the DOWN ARROW key until the X AXIS field is highlighted.



Press PROBE soft key.



Press EDGE soft key.

D:0 | T:1 | F: 0 | 0:00 | MM | ABS | | SET

SET DATUM		X	0.000	X			
DATUM NUMBER		Y	0.000				
0		Z	0.000				
DATUM		Select the probe function.		Y			
X							
Y							
Z				Z			
EDGE		CENTER LINE		CIRCLE CENTER		HELP	



PROBE IN X



Touch workpiece edge.

NOTE

Store the position of the edge by pressing the NOTE soft key. The NOTE soft key is useful when determining tool data by touching the workpiece in the absence of an edge finder with feedback. To avoid losing the position value when the tool is retracted, press the NOTE soft key to store the value while it 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 NOTE soft key.



Retract the tool from the workpiece.

ENTER VALUE FOR X

0

Enter coordinate of the edge

and

Enter

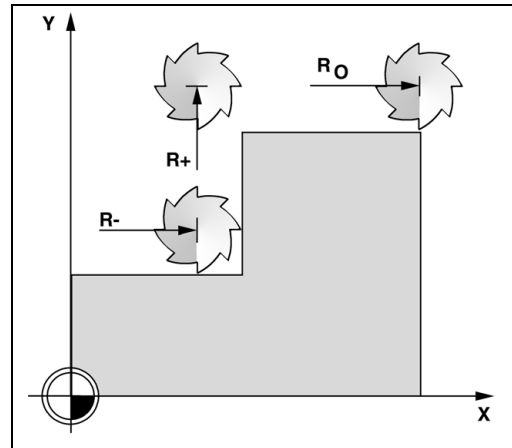
press **Enter**.



Presetting

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 Incremental 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 incremental value to account for the diameter of the tool.

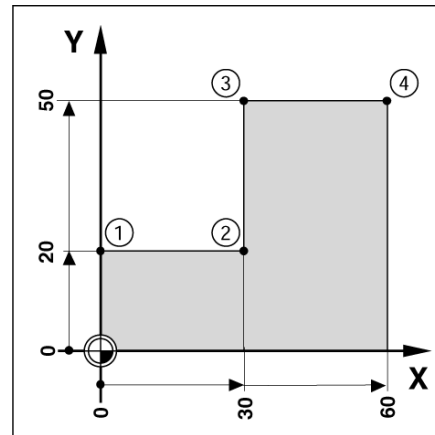



Absolute Distance Preset

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.

- Corner 1: X = 0 mm / Y = 20 mm
- Corner 2: X = 30 mm / Y = 20 mm
- Corner 3: X = 30 mm / Y = 50 mm
- Corner 4: X = 60 mm / Y = 50 mm



 To recall the last entered preset for a particular axis, press the axis key.



Preparation:

- ▶ Select the tool with the appropriate tool data.
- ▶ Pre-position the tool to an appropriate location (such as X = Y = -20 mm).
- ▶ Move the tool to milling depth.



Press the SET/ZERO soft key to be in the Set mode.

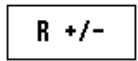


Press the Y axis key

NOMINAL POSITION VALUE



Enter nominal position value for corner point **1**:
Y = 20 mm, and



select tool radius compensation R + with R+/- soft key. Press until R+ is shown next to axis form.



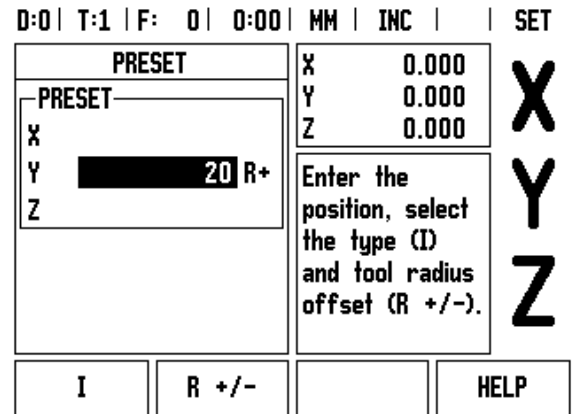
Press **Enter**.



Traverse the Y axis until the display value is zero. The square in the graphic positioning aid is now centered between the two triangular marks.



Press the X axis key



NOMINAL POSITION VALUE

3 0

Enter nominal position value for corner point **2**:
X = +30 mm,

R +/-

select tool radius compensation R – with R+/- soft key. Press twice until R+ is shown next to axis form.

Enter

Press **Enter**.



Traverse the X axis until the display value is zero. The square in the graphic positioning aid is now centered between the two triangular marks.



Press the Y axis key



NOMINAL POSITION VALUE

5 0

Enter nominal position value for corner point **3**:
Y = +50 mm,

R +/-

select tool radius compensation R + with R+/- soft key, and press until R+ is shown next to axis form.

Enter

Press **Enter**.



Traverse the Y axis until the display value is zero. The square in the graphic positioning aid is now centered between the two triangular marks.



Press the X axis key

NOMINAL POSITION VALUE

6 0

Enter nominal position value for corner point **4**:
X = +60 mm,

R +/-

select tool radius compensation R +, and press **Enter**.



Traverse the X axis until the display value is zero. The square in the graphic positioning aid is now centered between the two triangular marks.

Incremental Distance Preset

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

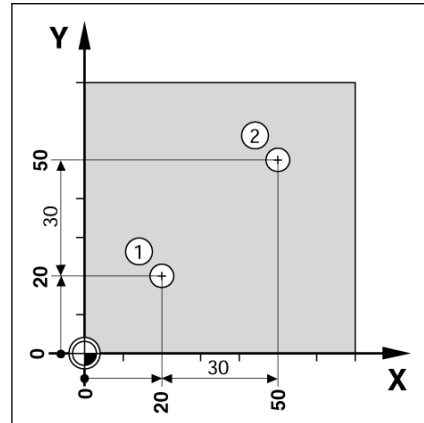
Enter the coordinates in incremental dimensions. These are indicated in the following (and on the screen) with a preceding **I**. The datum is the workpiece zero.

Hole **1** at: X = 20 mm / Y = 20 mm

Distance from hole **2** to hole **1**: XI = 30 mm / YI = 30 mm

Hole depth: Z = -12 mm

Operating mode: INCREMENTAL



Press the SET/ZERO soft key to be in the Set mode.



Press the X axis key.

- NOMINAL POSITION VALUE -



Enter nominal position value for hole 1: X = 20 mm, and ensure no tool radius compensation is active.



Press the DOWN ARROW key.

NOMINAL POSITION VALUE



Enter nominal position value for hole 1: Y = 20 mm. Ensure no tool radius compensation is showing.



Press the DOWN ARROW key.

NOMINAL POSITION VALUE



Enter the nominal position value for the hole depth: Z = -12 mm. Press **Enter**.



Drill hole **1**: Traverse the X, Y, and Z axis until the display value is zero. The square in the graphic positioning aid is now centered between the two triangular marks.

Retract the drill.



Press the X axis key



NOMINAL POSITION VALUE

3 0

Enter nominal position value for hole **2**: X = 30 mm,

I

mark the input as an incremental dimension, press I soft key.

Enter

Press **Enter**.



Press the Y axis key.

NOMINAL POSITION VALUE

3 0

Enter nominal position value for hole **2**: Y = 30 mm,

I

mark the input as an incremental dimension, press I soft key.

Enter

Press **Enter**.



Traverse the X, and Y axes until the display value is zero. The square in the graphic positioning aid is now centered between the two triangular marks.



Press the Z axis key.

NOMINAL POSITION VALUE

Enter

Press **Enter** (uses last entered preset).



Drill hole **2**: Traverse Z axis until the display value is zero. The square in the graphic positioning aid is now centered between the two triangular marks. Retract the drill.

D:0 T:1 F: 0 0:00 MM INC SET	
PRESET	
X	0.000
Y	0.000
Z	0.000
Enter the position, select the type (I) and tool radius offset (R +/-).	
I	R +/-
HELP	

X

Y

Z



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 the 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.

D:0 T:1 F: 0 0:00 MM INC			
RPM CALCULATOR			
DIAMETER			
5.0000 MM			
SURFACE SPEED			
47.1239 M/MIN			
SPINDLE SPEED			
3000.0 RPM			
UNITS			HELP

Enter the diameter of the rotating tool or part. The spindle speed will be calculated.



Circle, and linear Patterns (Milling)

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

Press the circle pattern, or linear pattern hard keys to select the desired hole pattern function, and enter the required data. This data can usually be taken from the workpiece drawing (e.g. hole depth, number of holes, etc.).

With hole patterns, the ND 522/523 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 machining starts. 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.	VIEW
Press to go to previous hole.	PREVIOUS HOLE
Press to manually advance to the next hole.	NEXT HOLE
Press this to use the existing position.	NOTE
Press to end drilling.	END

Circle Pattern

Information required:

- Pattern type (full, or segment)
- Holes (number of)
- Center (center of circle pattern in pattern plane)
- Radius (defines radius of the circle pattern)
- Start angle (angle of 1st hole in the pattern) - The start angle is between the zero angle reference axis, and the first hole (for added information see page 14).
- Step angle (optional: this only applies if creating a circle segment.) - The step angle is the angle between holes.
- Depth (the target depth for drilling in the tool axis)

ND 522/523 calculates the coordinates of the holes which the tool can then be moved to by traversing to display value zero.



Example: Enter data, and execute a circle pattern.

Holes (no. of): 4

Coordinates of center: X = 10 mm / Y = 15 mm

Bolt circle radius: 5 mm

Start angle: (Angle between X axis, and 1st hole): 25°

Hole depth: Z = -5mm

1st step: Enter data



Press CIRCLE PATTERN hard key.

PATTERN TYPE



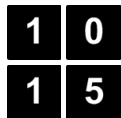
Enter the type of circle pattern (full). Cursor to the next field.

HOLES



Enter the number of holes (4).

CIRCLE CENTER



Enter the X, and Y coordinates of the circle center
Example: (X = 10), (Y = 15), or press NOTE to set the coordinate to the current position. Cursor to the next field.

RADIUS



Enter the radius of the circle pattern (5).

START ANGLE



Enter the start angle (25°).

STEP ANGLE



Enter the step angle (90°) (this can only be changed if entering a "segment").

D:0 | T:1 | F: 0 | 0:00 | MM | INC |

CIRCLE PATTERN		Enter the coordinates of the circle center.
TYPE	FULL	
HOLES	4	
CENTER	X 10.000 Y 15	
NOTE		HELP

D:0 | T:1 | F: 0 | 0:00 | MM | INC |

CIRCLE PATTERN		Set the pattern direction by pressing the - key.
RADIUS	5.000	
START ANGLE	25.000°	
STEP ANGLE	90.000°	
NOTE		HELP

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

	X	-14.530	
	Y	-17.115	
	Z	0.000	
Move (X,Y) to 0.0 then move Z to 0.0.			
VIEW	PREVIOUS HOLE	NEXT HOLE	END



DEPTH



Enter the depth when needed. The depth of the hole is optional, and may be left blank.



Press **Enter**.



Pressing the VIEW soft key will toggle between the three views of the pattern (the Graphic, DTG, and Absolute).

2nd step: Drill



Move to hole:

Traverse the X, and Y axes until display value zero.



Drill:

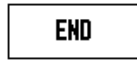
Traverse to display value zero in the tool axis.



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 pattern is complete, press the END soft key.



Linear Pattern

Information required:

- 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)



Example: Enter data, and execute a linear pattern.

Type of pattern: Array

First X coordinate of hole: X = 20 mm

First Y coordinate of hole: Y = 15 mm

Number of holes per row: 4

Hole spacing: 10 mm

Tilt angle: 18°

Hole depth: -2

Number of rows: 3

Row spacing: 12 mm

1st step: Enter data



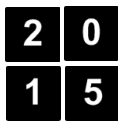
Press LINEAR PATTERN hard key.

PATTERN TYPE



Enter the type of pattern (Array). Cursor to the next field.

FIRST HOLE X, AND Y



Enter the X, and Y coordinates (X = 20), (Y = 15).
Cursor to the next field.

HOLES PER ROW



Enter the number of holes per row (4). Cursor to the next field.

HOLE SPACING

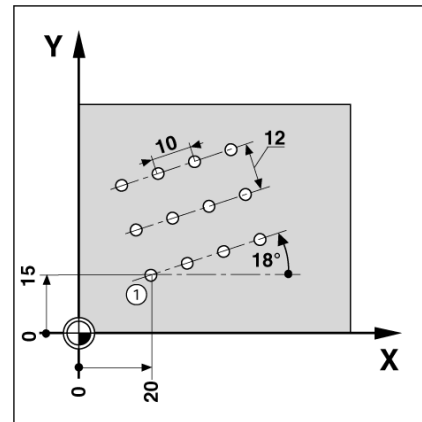


Enter the hole spacing (10).

ANGLE



Enter the tilt angle (18°).



D:0 | T:1 | F: 0 | 0:00 | MM | INC |

LINEAR PATTERN		Select the pattern type (ARRAY or FRAME).
TYPE	ARRAY	
FIRST HOLE		
X	20.000	
Y	15.000	
HOLES PER ROW	4	
ARRAY FRAME		HELP

D:0 | T:1 | F: 0 | 0:00 | MM | INC | H:1/12

	X -20.000 Y -15.000 Z 2.000		
	Move (X,Y) to 0.0 then move Z to 0.0.		
VIEW	PREVIOUS HOLE	NEXT HOLE	END



DEPTH

- 2

Enter the depth when needed (-2). The depth of the hole is optional, and may be left blank.

NUMBER OF ROWS

3

Enter the number of rows (3).

ROW SPACING

1 2

Enter the spacing between rows, press **Enter**.

Enter

VIEW

Pressing the VIEW soft key to see the graphic.

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.

NEXT HOLE

Press the NEXT HOLE soft key.

END

Continue to drill the remaining holes in the same way. When pattern is complete, press the END soft key.







Incline & Arc Milling

This section describes the functions for Incline, and Arc milling features.

By pressing either the Incline milling hard key, or the Arc milling hard key, will open the associated Entry Form. These features provide ways to machine a flat diagonal surface (incline milling), or a rounded surface (arc milling) using a manual machine.

Functions for Incline, and Arc Milling.

Function	Soft key
Press this to select a plane.	
Press this to use the existing position.	
Press to return to the previous step.	
Press to advance to the next step.	

Incline Milling

Entry Form:

- 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 NOTE to set the coordinate to the current position.
- End Point: Enter the coordinates of the end point, or press NOTE 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** 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 incline milling form, and pressing the **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.
- 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.



Example: Press the Incline Milling hard key to open the Form:

Plane: XY (3 choices are available- XY, YZ, & XZ) Select appropriate plane.

Start Point: Enter data, or press Note soft key

1st step: Enter data

PLANE [XY] Press PLANE soft key to select the milling plane.

 Press DOWN ARROW hard key.

START POINT

Enter the coordinates of the first axis start point, or press note to set the coordinate to current position.
Press DOWN ARROW hard key
Enter the coordinates of the second axis start point, or press note to set the coordinate to current position

NEXT DATA ENTRY

 Press DOWN ARROW hard key

END POINT

Enter the coordinates of the first axis end point, or press note to set the coordinate to current position.
Press DOWN ARROW hard key
Enter the coordinates of the second axis end point, or press note to set the coordinate to current position


NEXT DATA ENTRY

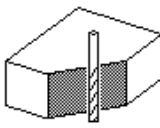
 Press DOWN ARROW hard key


STEP SIZE

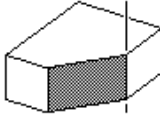
Enter the step size. The Step size is optional. If the value is zero, the operator decides at run-time how far to move between each pass.

Enter press **Enter** to run the program, or the end soft key to exit.

D:0 | T:1 | F: 0 | 0:00 | MM | ABS | 

MILL LINE		Press PLANE to select the plane.
PLANE	XY	
START POINT		
X	0.000	
Y	0.000	
PLANE [XY]		HELP

D:0 | T:1 | F: 0 | 0:00 | MM | ABS | 

MILL LINE		Enter the END point.
END POINT		
X	0.000	
Y	0.000	
STEP		
0.000		
NOTE		HELP

Arc Milling

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.
- **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.



- The contour view show 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.



Example: Press the Arc Milling hard key to open the Entry Form:

Plane: XY (3 choices are available- XY, YZ, & XZ) Select appropriate plane.

Center Point: Enter data, or press Note soft key

1st step: Enter data



Press PLANE soft key to select the milling plane.



Press DOWN ARROW hard key.

CENTER POINT

Enter the coordinates of the center point, or press NOTE to set the coordinate to current position. Press DOWN ARROW hard key

NEXT DATA ENTRY



Press DOWN ARROW hard key

START, AND END POINT

Enter the XY coordinates of the axis start point, or press NOTE to set the coordinate to current position. Enter the coordinates of the axis end point, or press NOTE. Press DOWN ARROW hard key

NEXT DATA ENTRY



Press DOWN ARROW hard key

STEP SIZE

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



press **Enter** to run the program, or the END soft key to exit.

D:0 | T:1 | F: 0 | 0:00 | MM | ABS |

MILL ARC		Press PLANE to select the plane.					
PLANE							
XY							
CENTER POINT							
X	0.000						
Y	0.000	<table border="1"> <tr> <td>PLANE [XY]</td> <td></td> <td>RUN</td> <td>HELP</td> </tr> </table>		PLANE [XY]		RUN	HELP
PLANE [XY]		RUN	HELP				

D:0 | T:1 | F: 0 | 0:00 | MM | ABS |

MILL ARC		Enter the START point.					
START POINT							
X	0.000						
Y	0.000						
END POINT							
X	0.000	<table border="1"> <tr> <td>NOTE</td> <td></td> <td></td> <td>HELP</td> </tr> </table>		NOTE			HELP
NOTE			HELP				
Y	0.000						

D:0 | T:1 | F: 0 | 0:00 | MM | ABS |

MILL ARC		Enter the step size.					
STEP							
0.000							
				<table border="1"> <tr> <td></td> <td></td> <td></td> <td>HELP</td> </tr> </table>			
			HELP				



I - 4 Turning Specific Operations

This section discusses operations, and key functions specific to turning applications only. Key functions that are the same, whether the ND 522/523 is configured for Mill, or Turn applications, are detailed starting on page 17.

Keys Functions Detailed

Turning Specific Display Icons

Function	Display Icon
This is used to indicate that the displayed value is a diameter value. No icon visible indicates that the display is a radius value.	∅

Tool Hard key

The ND 522/523 can store the dimensional offsets for up to 16 tools. When a workpiece is changed, and a new datum is established, all tools are automatically referenced from the new datum.

Before using a tool, its offset must be entered (the cutting edge position). Tool offsets can be set using the TOOL/SET, or NOTE/SET features.

If tools have been measured using a tool presetter, the offsets may be entered directly.

To access the Tool Table menu:



Press the TOOL hard key.

The cursor will default to the TOOL TABLE field.

TOOL TABLE



Scroll to the tool to be defined, and press **Enter**.

D:0 | T:1 | F: 0 | 0:00 | MM | ABS | |

TOOL TABLE (X/Z)	
1	19.082∅
2	
3	
4	
5	19.451∅
6	
7	
8	

CLEAR
TOOL

USE
TOOL

HELP



Tool Table Usage

Example: Entering offsets into the tool table

Setting tool offsets 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.



Press the TOOL hard key. Scroll to the desired tool.

Enter

Press the **Enter** key.



Select the axis (X) key.

2 0

Enter the position of the tool tip, for example, X=Ø 20 mm.

Remember to ensure the ND 522/523 is in diameter display mode (Ø) to input a diameter value.

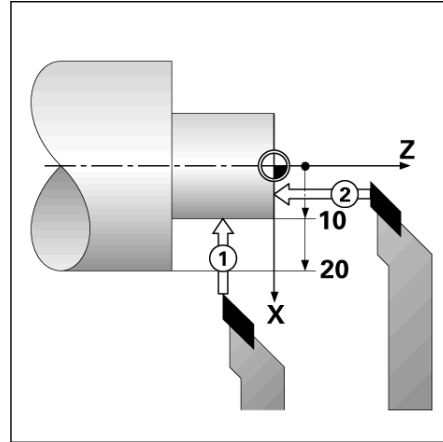
Touch the workpiece face with the tool.



Cursor to the Z axis.

0

Set the position display for the tool tip to zero, Z=0. Press **Enter**.










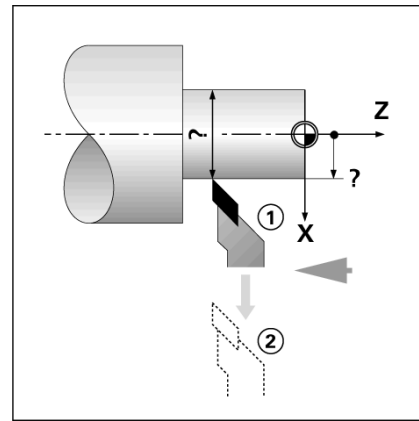
Setting Tool Offset using NOTE/SET Function

The NOTE/SET 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 NOTE/SET 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 NOTE.

To use the NOTE/SET function:

-  Press the TOOL hard key. Select the desired tool, and press **Enter** key.
 -  Select the X axis key.
 -  Turn a diameter in the X axis.
 -  Press the NOTE 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, for example, 15 mm, and press **Enter**.
- Remember to ensure the ND 522/523 is in diameter display mode (Ø) to input a diameter value.



D:0 | T:1 | F: 0 | 0:00 | MM | ABS | | SET

TOOL/SET		X	0.000Ø	X Z
TOOL		Z	0.000	
X	15.000 Ø	Turn dia. in X then press NOTE or enter tool position.		
Z				
NOTE				HELP



Datum Hard key

See "Datum Hard key" on page 35 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.

Axis sequence in this example: X - Z

Preparation:

Call the tool data by selecting the tool which being used to touch the workpiece.



Press the DATUM hard key.

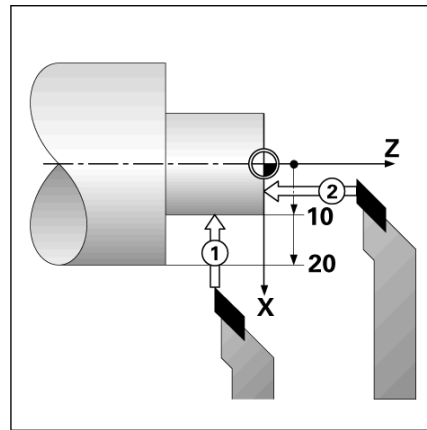
The cursor will be in the DATUM NUMBER field.



Enter the datum number, and press the DOWN ARROW key to go to the X AXIS field.



Touch the workpiece at point **1**.



D:2 T:1 F: 0 0:00 MM ABS SET	
SET DATUM	
DATUM NUMBER	2
DATUM	
X	20.000 ∅
Z	0.0
Face part in Z then press NOTE or enter tool position.	
NOTE	HELP

X
Z



DATUM SETTING X

2 0

Enter the diameter of the workpiece at that point.



Remember to ensure the ND 522/523 is in diameter display mode (\varnothing) to input a diameter value.

Press the DOWN ARROW key to advance to the Z-axis.



Touch the workpiece surface at point 2.

DATUM SETTING Z

0

Enter the position of the tool tip ($Z = 0$ mm) for the Z-coordinate of the datum.

Enter

Press **Enter**.



Setting Datums using NOTE/SET Function

The NOTE/SET 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 NOTE/SET 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.

NOTE

Press the NOTE soft key while the tool is still cutting.

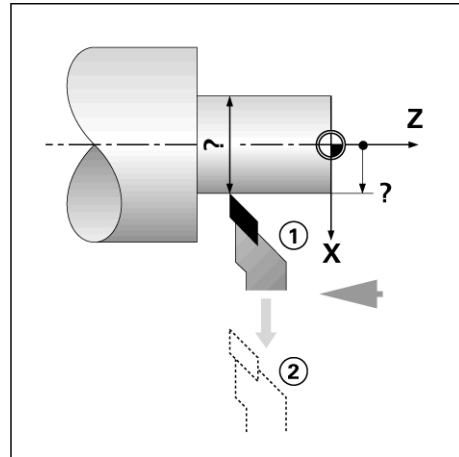
Retract from the current position.

Turn the spindle off, and measure the workpiece diameter.

1 5

Enter the measured diameter, for example, 15 mm, and press **Enter**.

Remember to ensure the ND 522/523 is in diameter display mode (Ø) to input a diameter value.



D:2 | T:1 | F: 0 | 0:00 | MM | ABS | SET

SET DATUM		X	0.000 \emptyset	X Z
DATUM NUMBER		Z	0.000	
2		Enter the new actual position of the tool.		
DATUM				
X	15 \emptyset			
Z				
				HELP



Taper Calculator Hard Key

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.

Entry values:

For the taper ratio, calculation requires:

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

For taper calculations 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 has now changed, and includes the taper calculator functions.

D1/D2 LENGTH

**TAPER:
D1/D2/L**

To calculate the taper angle using two diameters, and length between, press the TAPER: D1/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

TAPER RATIO

**TAPER:
RATIO**

To calculate angles using the ratio of the diameter change to length, 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.

D:0 | T:1 | F: 0 | 0:00 | MM | ABS | |

TAPER CALCULATOR		Enter second diameter.
DIAMETER		
D1	10.0000	
D2	5.0000	
LENGTH		
ANGLE		
5.7106°		
NOTE		HELP

D:0 | T:1 | F: 0 | 0:00 | MM | ABS | |

TAPER CALCULATOR		Enter first diameter.
DIAMETER		
D1	10.0000	
D2	5.0000	
LENGTH		
ANGLE		
5.7106°		
NOTE		HELP

Presetting

The Preset function has been explained previously in this manual (See "Presetting" on page 39). 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.

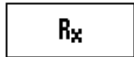
While doing turning, input values can be either radius, or diameter values. It is important to be sure the units being entered for the preset agree with the state that the display is currently using. A diameter value is shown with a \varnothing symbol. The state of the display can be changed using the RX soft key (see below).

RX (Radius/Diameter) Soft Key

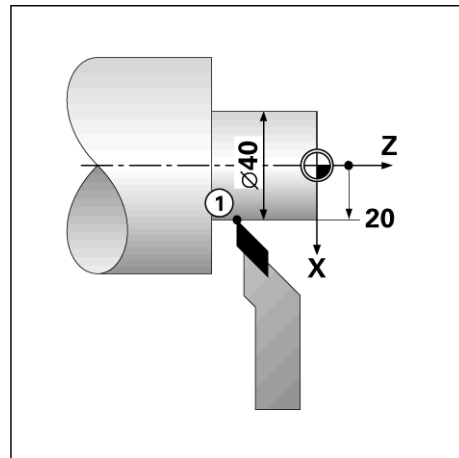
Drawings for lathe parts usually give diameter values. ND 522/523 can display either the radius, or the diameter for you. When the diameter is being displayed, the diameter symbol (\varnothing) is shown next to the position value.

Example: Radius display, position 1 X = 20 mm

Diameter display, position 1 X = \varnothing 40 mm



Press the RX soft key to switch between radius display, and diameter display.



Vectoring Hard Key

Vectoring breaks down the movement of the compound axis into the crossfeed, or longitudinal axes. When turning threads, for example, vectoring displays the diameter of the thread in the X-axis display, even though the cutting tool is being moved with the compound axis handwheel. With vectoring enabled, the desired radius can be preset, or diameter in the X-axis, so that it is possible to “machine to zero”.

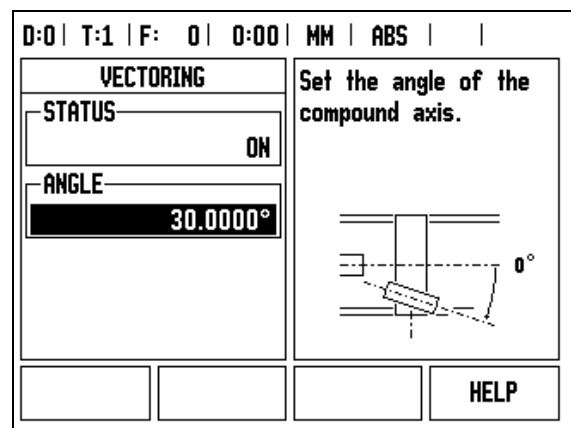


When vectoring is used, the top slide (compound) axis encoder must be assigned to the bottom display axis. The crossfeed component of movement of the axis will then be shown in the top display axis. The longitudinal component of movement of the axis will be shown in the middle display axis.

Press the VECTORING hard key.

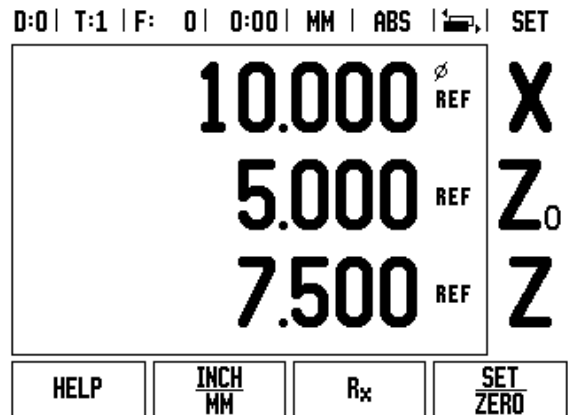
Press the ON soft key to enable the vectoring feature.

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**.



Z Coupling (turning applications only)

The ND 522/523 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.



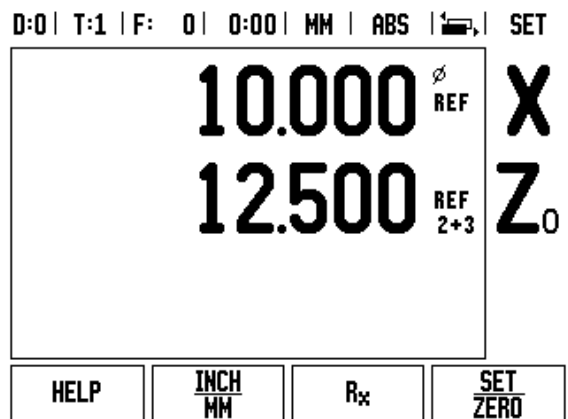
Enabling Z Coupling

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.

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.

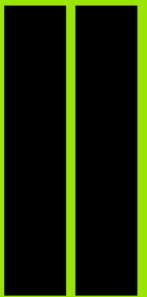
When a position is coupled, the reference mark for both encoders must be found in order to recall the previous datum.



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.





Technical Information



II - 1 Installation, and Electrical Connection

Items Supplied

- ND 522/523 Display Unit
- Power connector
- Quick Reference Guide
- Tilt / Swivel Assembly

Accessories

- Mounting base
- Assorted Mounting Arm assemblies

ND 522/523 Display Unit

Mounting Location

Locate the unit in a well ventilated area such that it may be easily accessed during normal operation.

Installation

A locking handle is used to secure the ND 522/523 from below on to a mounting arm. The DRO mounting assembly comes complete with swivel / tilt Kit: See "ND 522/523 Handle ID 618025-01" on page 89.

Electrical connection



There are no serviceable items within this unit. Therefore, the ND 522/523 must never be opened.

The length of the power cord is not to exceed 3 meters.

Connect a protective ground to the protective conductor terminal on the rear of the unit. This connection must never be interrupted.



Do not engage, or disengage any connections while the unit is under power. Damage to internal components may result.

Use only original replacement fuses.



Electrical requirements

Voltage AC 100 V ... 240 V (-15 % ... +10 %)

Power Max. 54 W

Frequency 47 Hz ... 63 Hz (± 3 Hz)

FuseT 500 mA/250 V, 5 mm x 20 mm, Slo-Blo (line, and neutral fused)

Environmental

Protection (EN 60529) IP 40 back panel

IP 54 front panel


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 (5.8 lb.)

Wiring the power connector

Hot leads: L, and N

Ground: 

Minimum diameter of power connection cable: 0.75 mm².

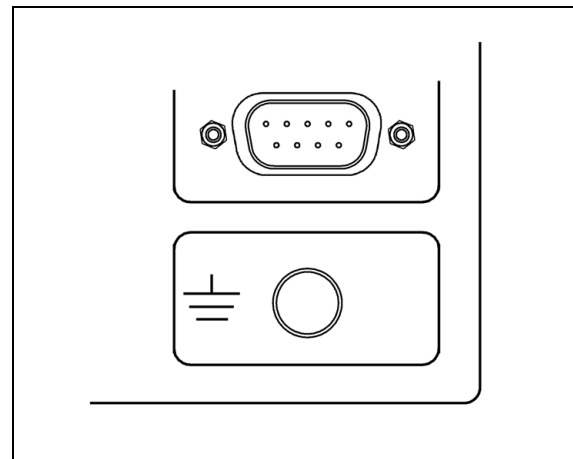
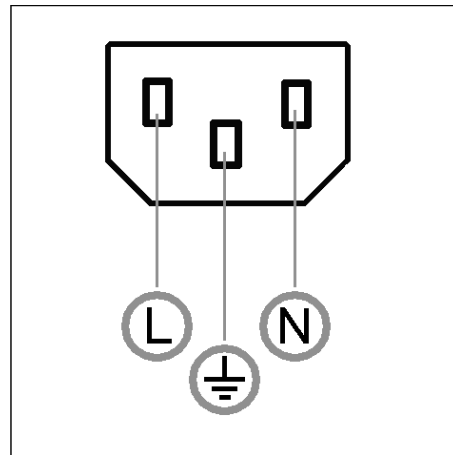
Protective earthing (grounding)



It is necessary to connect the protective ground terminal on the rear panel to the star point of machine ground.
Minimum cross-section of the connecting wire: 6 mm².

Preventative maintenance

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



Connecting the Encoders

ND 522/523 can be used with **HEIDENHAIN** linear, and rotary encoders that provide digital TTL level signals.

The **connecting cable** must not exceed 30 m (100 ft.) in length.



Do not engage, or disengage any connections while the unit is under power.

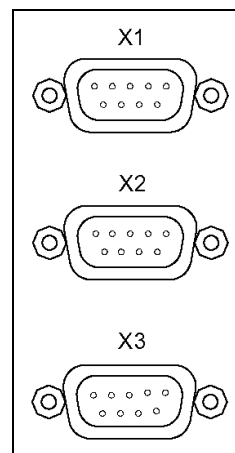
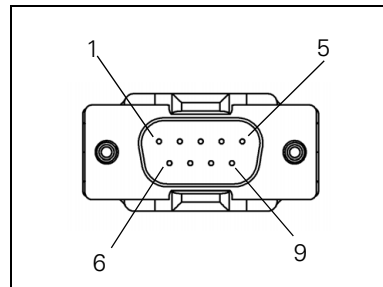
Pin layout for encoder inputs.

D-sub connector 9-pin	Signal
1	/
2	U_{a1}
3	$\overline{U_{a1}}$
4	U_{a2}
5	$\overline{U_{a2}}$
6	0 V
7	U_P
8	U_{a0}
9	$\overline{U_{a0}}$

The operator can setup any encoder input to go to any axis.

Default configuration:

Encoder input	Mill	Turn
X1	X	X
X2	Y	Z_0
X3	Z	Z



II - 2 Installation Setup

Installation Setup Parameters

Installation setup is accessed by pressing the SETUP soft key, which brings up INSTALLATION SETUP soft key.

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 a passcode: (95148). Type these digits in using the numeric key pad, and then press **Enter**.

D:0 T:1 F: 0 0:00 MM ABS			
INSTALLATION SETUP ENCODER SETUP DISPLAY CFG. ERROR COMPENSATION BACKLASH COMP. COUNTER SETTINGS DIAGNOSTICS	Set the counter application (MILL or TURN) and number of axes.		
JOB SETUP	IMPORT EXPORT		HELP

Encoder Setup

The Encoder Setup form is used to set the encoder resolution, and type (linear, rotary), count direction, reference mark type.

- ▶ 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 to be changed, 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 lines per revolution.
- ▶ In the REFERENCE MARK field, toggling the REF MARK soft key selects whether the encoder has no reference signal (NONE), single reference mark (SINGLE), or (CODED) soft key.
- ▶ For coded reference marks, press the SPACING soft key to select 500, 1000 or 2000 (LB 382C).
- ▶ 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.

D:0 T:1 F: 0 0:00 MM ABS			
ENCODER SETUP (1) ENCODER TYPE LINEAR RESOLUTION 5.0 μm REFERENCE MARK CODED / 1000	Press REF MARK to select the type of reference marks on the encoder. CODED / 2000 is LB 382C only.		
REF MARK [CODED]	SPACING [1000]		HELP



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.
- ▶ Press 1, 2, or 3 to select the encoder input number (X1, X2, or X3).
- ▶ Scroll to the DISPLAY RESOLUTION field. Press the COARSER, or FINER soft keys to select the display resolution.
- ▶ For rotary encoders, scroll to the angle display field. Press ANGLE soft key to select the format for the angle display.

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.

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. These errors can be determined with a reference measurement system, e.g. the VM 101 from HEIDENHAIN, or with gauge blocks. From an analysis of the error it can be determined which form of compensation is required, linear, or non-linear error.

The ND 522/523 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) / M) x 10⁶ ppm** with:

- S** measured length with reference standard
- M** measured length with device at axis

Example:

If the length of the standard 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) / 499.95) x 10⁶ ppm = 100 ppm (rounded to the nearest whole number).

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

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. ND 522/523 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. Press enter.
- ▶ 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, move to the location of the start point, and press TEACH POSITION. Press **Enter**.

D:0 T:1 F: 0 0:00 MM ABS									
<table border="1" style="width: 100%; border-collapse: collapse;"> <tr> <th colspan="2" style="text-align: center;">ERROR COMPENSATION</th> </tr> <tr> <td style="padding: 2px;">INPUT X1</td> <td style="text-align: center; padding: 2px;">0 PPM</td> </tr> <tr> <td style="padding: 2px;">INPUT X2</td> <td style="text-align: center; padding: 2px;">OFF</td> </tr> <tr> <td style="padding: 2px;">INPUT X3</td> <td style="text-align: center; padding: 2px;">OFF</td> </tr> </table>	ERROR COMPENSATION		INPUT X1	0 PPM	INPUT X2	OFF	INPUT X3	OFF	Error compensation for this input is OFF. Press TYPE to select linear or non-linear error compensation.
ERROR COMPENSATION									
INPUT X1	0 PPM								
INPUT X2	OFF								
INPUT X3	OFF								
TYPE [OFF]	HELP								



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.



Pressing **Enter** saves the Error Compensation data entered, and is retained if the power is cycled. Pressing the C key to exit without pressing the **Enter** key only retains data until power is cycled.

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.

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 0:00 MM ABS			
BACKLASH COMP.		Specify the amount of backlash between the encoder and the machine.	
INPUT X1	0.2		
INPUT X2	OFF		
INPUT X3	OFF		
ON OFF			HELP



Counter Settings

The COUNTER SETTINGS form 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. 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. An axes soft key will appear to choose between either 2, or 3 axes.

D:0 | T:1 | F: 0 | 0:00 | MM | 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	FACTORY DEFAULTS	HELP

Diagnostics

The DIAGNOSTICS form provides access for testing the keypad, and display.

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 then back to normal.

D:0 | T:1 | F: 0 | 0:00 | MM | ABS | |

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



II - 3 Encoder Parameters

The following tables represent a partial list of encoders. These tables describe all operating parameters that must be set for the encoders. Most entries can be found in the operating instructions for that encoder.

Example settings for HEIDENHAIN linear encoders

Encoder	Signal period	Reference marks
LS 328C LS 628C	5 μ m	Single/1000
LB 382 with IBV 101	1 μ m	Single
LB 382C with IBV 101	1 μ m	Single/2000
LS 378C LS 678C	1 μ m	Single/1000

Example settings for HEIDENHAIN Rotary encoders

Rotary Encoders	Line Count	Resolution	Reference mark
ROD 420	50 to 5000	1.8° to 64.8 arc sec	single
ROD 426	50 to 10000	1.8° to 32.4 arc sec	single
ROD 1020	250 to 3600	.36° to 90 arc sec	single
ROD 1070	1000 to 3600	32.4 to 9 arc sec	single
ERN 120	1000 to 5000	324 to 64.8 arc sec	single
ERN 420	250 to 5000	.36° to 64.8 arc sec	single
ERN 1020	250 to 3600	.36° to 90 arc sec	single
ERN 1070	1000 to 3600	32.4 to 9 arc sec	single

Example settings for HEIDENHAIN Angle encoders

Angle Encoders	Line Count	Resolution	Reference mark
ROD 225	9000, or 10000	36 to 18 arc sec	single
ROD 275	18000	3.6 to 1.8 arc sec	single



II - 4 Data Interface

The ND 522/523's data interface includes the USB port. The USB port supports both bi-directional data communications which allows data to be exported to, or imported from an external device, and external operations via the data interface.

Data which can be exported from the ND 522/523 to an external serial device includes:

- Job, and installation configuration parameters
- Non-linear error compensation tables

Data which can be imported into ND 522/523 from an external device includes:

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

This chapter covers information about **setting up** the data interface:



USB Port (type “B”)

The USB port is located on the rear panel. The following device can be connected to this port:

- Personal computer with serial data interface

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

To export, or import data between the ND 522/523, and a personal computer, the PC must use terminal communications software such as TNC Remo. (TNC Remo is available for free at: http://filebase.heidenhain.de/doku/english/serv_0.htm. Contact a Heidenhain distributor for further details.) This software processes the data being sent, or received over the serial cable link. All data transferred between ND 522/523, and the PC is in ASCII text format.

To export data from the ND 522/523 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 ND 522/523's IMPORT/EXPORT soft key. Select Export.

To import data into the ND 522/523 from a PC, the ND 522/523 must first be made ready to receive the data. Press the ND 522/523's IMPORT/EXPORT soft key. Select Import. After the ND 522/523 is ready, setup the terminal communications program on the PC to send the desired file in ASCII text format.

Data format

Data is transferred in the following sequence:



The ND 522/523 does not support communication protocols such as Kermit, or Xmodem.

External Operations via USB port

The display unit can be operated over the USB port data interface using an external device. The following key commands are available:

Format

<ESC>TXXXX<CR>	Key is pressed
----------------	----------------

Sequence of commands	Function
----------------------	----------

<ESC>T9000<CR>	Key '0'
----------------	---------

<ESC>T9001<CR>	Key '1'
----------------	---------

<ESC>T9002<CR>	Key '2'
----------------	---------

<ESC>T9003<CR>	Key '3'
----------------	---------

<ESC>T9004<CR>	Key '4'
----------------	---------

<ESC>T9005<CR>	Key '5'
----------------	---------



Sequence of commands	Function
<ESC>T9006<CR>	Key '6'
<ESC>T9007<CR>	Key '7'
<ESC>T9008<CR>	Key '8'
<ESC>T9009<CR>	Key '9'
<ESC>T9010<CR>	Key 'CE', or 'CL'
<ESC>T9011<CR>	Key '.'
<ESC>T9012<CR>	Key 'Enter'
<ESC>T9013<CR>	Key 'X'
<ESC>T9014<CR>	Key 'Y'/'Z'/'Z0'
<ESC>T9015<CR>	Key 'Z'
<ESC>T9016<CR>	Key 'Soft key 1'
<ESC>T9017<CR>	Key 'Soft key 2'
<ESC>T9018<CR>	Key 'Soft key 3'
<ESC>T9019<CR>	Key 'Soft key 4'
<ESC>T9020<CR>	Key 'Cursor Left'
<ESC>T9021<CR>	Key 'Cursor Right'
<ESC>T9022<CR>	Key 'Cursor Up'
<ESC>T9023<CR>	Key 'Cursor Down'
<ESC>T9024<CR>	Key '+'
<ESC>T9025<CR>	Key '-'
<ESC>T9026<CR>	Key 'x'
<ESC>T9027<CR>	Key '/'
<ESC>T9028<CR>	Key 'Square Root'
<ESC>T9029<CR>	Key 'Pi'
<ESC>T9030<CR>	Key 'INC/ABS'
<ESC>T9031<CR>	Key '1/2'
<ESC>T9032<CR>	Key 'Calc'
<ESC>T9033<CR>	Key 'Datum'
<ESC>T9034<CR>	Key 'Tool'



Sequence of commands	Function
<ESC>T9035<CR>	Key 'Circle Pattern'
<ESC>T9036<CR>	Key 'Linear Pattern'
<ESC>T9037<CR>	Key 'Incline Milling'/'Vectoring'
<ESC>T9038<CR>	Key 'Arc Mill/Taper Calc'



II - 5 Measured Value Output

Examples of character output at the data interface

A PC can be used to retrieve values from ND 522/523. In all three examples, measured value output is started with **Ctrl B** (sent over USB interface). **Ctrl B** will transmit the currently displayed values in either Incremental, or Absolute mode, whichever is currently visible.

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	11	12

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)



II - 6 Specifications for Milling

ND 522/523 Data	
Axes	2, or 3 axes from A - Z, 0-9
Encoder inputs	Digital TTL Signal levels; input frequency max. 100 kHz for incremental HEIDENHAIN encoders
Display step	Linear axes: 1 mm to 0.1 μ m Rotary axes: 1° to 0.0001° (00°00'01")
Display	<p>Monochrome display for position values, dialogue, and input display, graphic functions, graphic positioning aid</p> <ul style="list-style-type: none"> ■ Status display: Operating mode, REF, inch/mm, scaling factor, feed rate, stop watch Datum number Tool number Tool compensation R-, R+
Functions	<ul style="list-style-type: none"> ■ REF reference-mark evaluation for distance-coded, or single reference marks ■ Distance-to-go mode, nominal position input (Incremental, or Absolute) ■ Scaling factor ■ HELP: On-screen operating instructions ■ INFO: Calculator, cutting data calculator, user, and operating parameters ■ 10 datum points, and 16 tools ■ Tool radius compensation ■ Calculation of positions for bolt hole circles, and linear hole patterns
Error compensation	Linear, and non-linear, up to 200 measuring points
Backlash compensation	Rotary encoder applications with ballscrews
Data interface	<ul style="list-style-type: none"> ■ USB (type B connector) 115 200 baud For output of measured values, and parameters; For input of parameters, and remote keys
Accessories	<ul style="list-style-type: none"> ■ Mounting Base ■ Universal Mounting Arm Kits
Main power input	AC 100 V ... 240 V (-15 % ... +10 %); 47 Hz ... 63 Hz (\pm 3 Hz); power consumption max. 54 W
Operating temperature	0°C to 45°C (32 °F to 113 °F)
Storage temperature	-20°C to 70°C (-4°F to 158°F)
Degree of protection (EN 60529)	IP 40 (IP 54 front panel)
Weight	2.6 kg

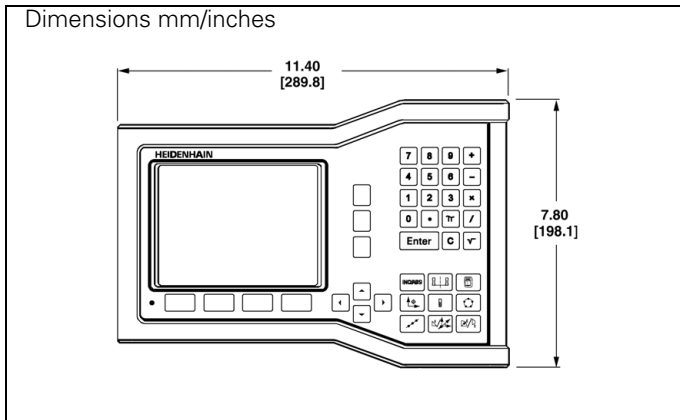


II - 7 Specifications for Turning

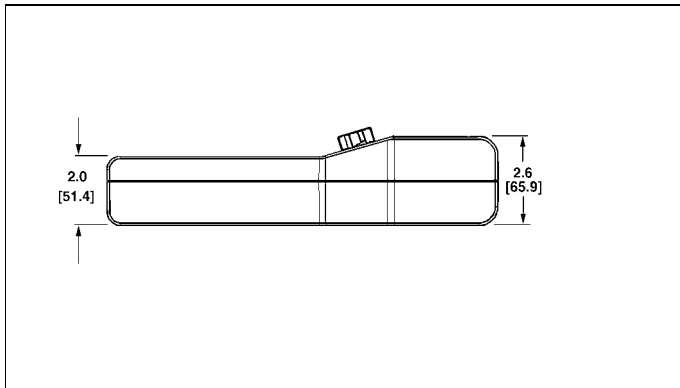
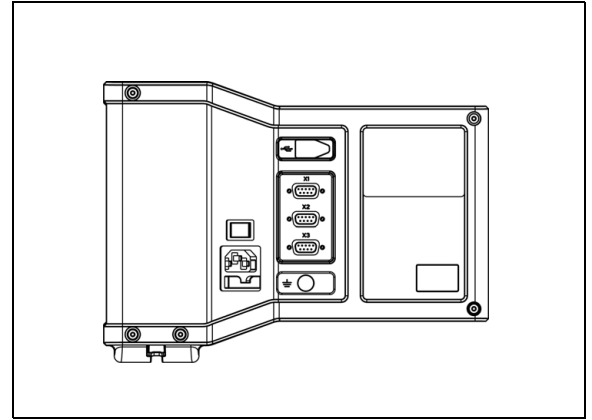
ND 522/523 Data	
Axes	2, or 3 axes from A to Z, Z0, 0-9
Encoder inputs	Digital TTL Signal levels; input frequency max. 100 kHz for incremental HEIDENHAIN encoders
Display step	Linear axes: 1 mm to 0.1 μm Rotary axes: 1° to 0.0001° (00°00'01")
Display	Monochrome display for position values, dialogue, and input display, graphic functions, graphic positioning aid ■ Status display: tool number, operating mode, REF, inch/mm, scaling factor, feed rate, diameter display \varnothing , stop watch, datum
Functions	<ul style="list-style-type: none"> ■ REF reference-mark evaluation for distance-coded, or single reference marks ■ Distance-to-go mode, nominal position input (Incremental, or Absolute) ■ Scaling factor ■ HELP: On-screen operating instructions ■ INFO: Calculator, taper calculator, user, and operating parameters ■ 10 datum point, 16 tools ■ Freezing tool position for back-off
Backlash Compensation	Rotary encoder applications with ballscrews
Error Compensation	Linear, and non-linear, up to 200 measuring points
Data interface	<ul style="list-style-type: none"> ■ USB (type B connector) 115 200 baud For output of measured values, and parameters; For input of parameters, remote keys, and commands
Accessories	<ul style="list-style-type: none"> ■ Mounting Base ■ Universal Mounting Arm Kits
Main power input	AC 100 V ... 240 V (-15 % ... +10 %); 47 Hz ... 63 Hz (± 3 Hz); power consumption max. 54 W
Operating temperature	0°C to 45°C (32 °F to 113 °F)
Storage temperature	-20°C to 70°C (-4°F to 158°F)
Degree of protection (EN 60529)	IP 40 (IP 54 front panel)
Weight	2.6 kg



II - 8 Dimensions



Front view with Dimensions



Bottom view with Dimensions

II - 9 Accessories

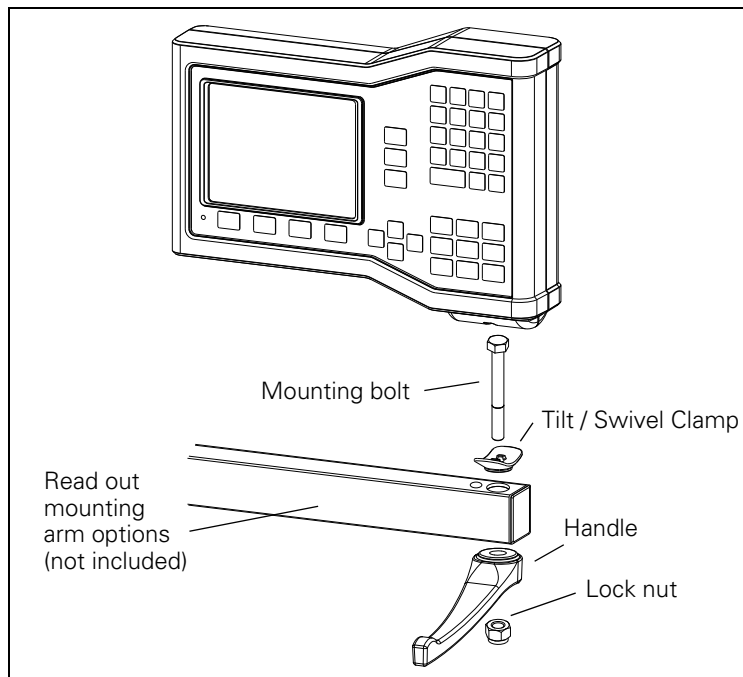
Accessory ID Numbers

ID Number	Accessories
532522-01	Pkgd, ND 522
532523-01	Pkgd, ND 523
625491-01	Pkgd, Mounting Base ND 522/523

ND 522/523 Handle

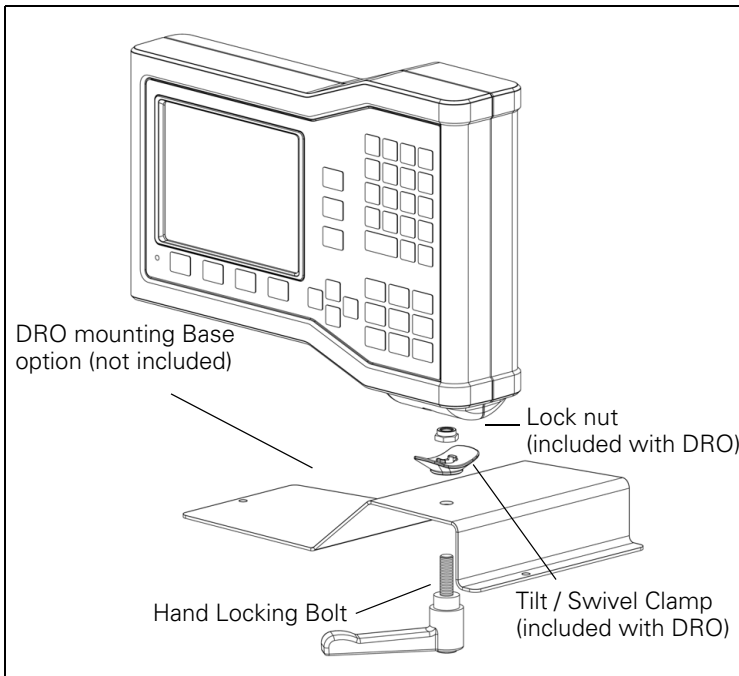
ID 618025-01

DRO mounting with arm (reference information)



The DRO mounting base has a slot feature that prevents the lock nut from turning. Pre-assemble all components to the mounting arm. Then slip the DRO on to the clamp. Secure the DRO into position by tightening the handle.

DRO mounting with arm (reference information)



The DRO mounting base accessory typically mounts to a flat surface on the machine. The mounting hardware assembly attaches to the DRO in the same fashion as the Handle assembly (which is included with the DRO).

SYMBOLE

1/2 soft key ... 29

A

Absolute coordinates ... 13
 Absolute datum ... 12
 Absolute Distance Preset ... 39
 Absolute Workpiece Positions ... 13
 Accessories ... 89
 Actual Value/Distance-To-Go soft key ... 18
 Arc Milling ... 55
 Axis Labels ... 16

B

Backlash Compensation ... 77

C

C (Clear) Key ... 18
 Calc soft key ... 30
 Centerline between two probed edges as datum ... 37
 Circle Pattern ... 46
 Connecting Cable ... 72
 Connecting the Encoders ... 72
 Console adjustment ... 26
 Counter Settings ... 67, 78

D

Data Input Forms ... 20
 Data Interface ... 80
 Datum setting without probing function ... 33, 36, 61
 Datum soft key ... 35
 Datum Soft Key (Turning) ... 61
 Datums ... 12
 Diagnostics ... 78
 Diameter Axes (Turning) ... 24
 Dimensions ... 88
 Disable Ref soft key ... 22
 Display Area ... 16
 Display configuration ... 74
 distance-coded reference marks ... 15

E

Electrical Connection ... 70
 Electrical Requirements ... 71
 Enable Ref soft key ... 21
 Enable/Disable Ref Function ... 22
 Encoder Parameters ... 79
 Encoder Setup ... 73
 Enter Key ... 18
 Environmental specs. ... 71
 Error Compensation ... 74
 Error Messages ... 20

F

fixed reference marks ... 15
 Fundamentals of Positioning ... 12

G

General Navigation ... 18
 General Operation's Soft Key Function Detailed ... 28
 General Operation's Soft Key Function Overview ... 17, 27
 Graphic Positioning Aid ... 19
 Graphic Positioning Aid (setting) ... 25

H

Handle ... 89
 Help Screen ... 19

I

Import/Export (setting) ... 26, 31
 Inch/MM soft key ... 23
 Incremental coordinates ... 13
 Incremental Distance Preset ... 43
 Incremental Workpiece Positions ... 13
 Index soft key ... 19
 Installation Setup Parameters ... 73
 Instruction Box ... 20
 Items Supplied ... 70

J

Job Setup menu ... 23
 Job Setup Parameters ... 23

K

Keypad, use ... 18

L

Language (setting) ... 26
 Left/Right arrow keys ... 18
 Linear Error Compensation ... 75
 Linear Pattern ... 49, 52

M

Measured Value Output ... 84
 Milling Specific Operations and Soft Key Functions Detailed ... 31
 Mirror ... 24
 Mounting Instructions for Mounting Arm ... 89
 Mounting Location ... 70

N

No Ref soft key ... 21
 Non-Linear Error Compensation ... 75

O

Operating Modes ... 18

P

Patterns (Milling) ... 46
 Position Encoders ... 14
 Position feedback ... 14
 Power Up ... 21
 Preset ... 39
 Preset Soft Key (Turning) ... 65
 Preventative Maintenance ... 71
 Protective earthing (grounding) ... 71

R

Radius/Diameter display ... 65
 REF ... 14
 Ref Symbols ... 16
 Reference Mark Evaluation ... 21
 Reference marks ... 15
 crossing over ... 21

S

Scale factor ... 24
 Screen Layout ... 16
 Set Zero soft key ... 28
 Setup ... 23
 Setup soft key ... 23
 Soft key Labels ... 16
 Specs. for Milling ... 86
 Specs. for Turning ... 87
 Status Bar (setting) ... 25
 Stopwatch (setting) ... 25

T

- Taper calculator ... 64
- Tool call ... 35
- Tool setting, turning ... 59
- Tool soft key ... 32
- Tool soft key (Turning) ... 58
- Tool Table Usage ... 33
- Tool Table Usage (Turning) ... 59

U

- Units of measurement, setting ... 23
- Up/Down arrow keys ... 18

V

- Vectoring ... 66

W

- Working without reference mark
evaluation ... 21

Z

- Z Coupling ... 67
- Zero Angle Reference Axis ... 14



HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH

Dr.-Johannes-Heidenhain-Straße 5

83301 Traunreut, Germany

☎ +49 8669 31-0

FAX +49 8669 5061

E-mail: info@heidenhain.de

www.heidenhain.de