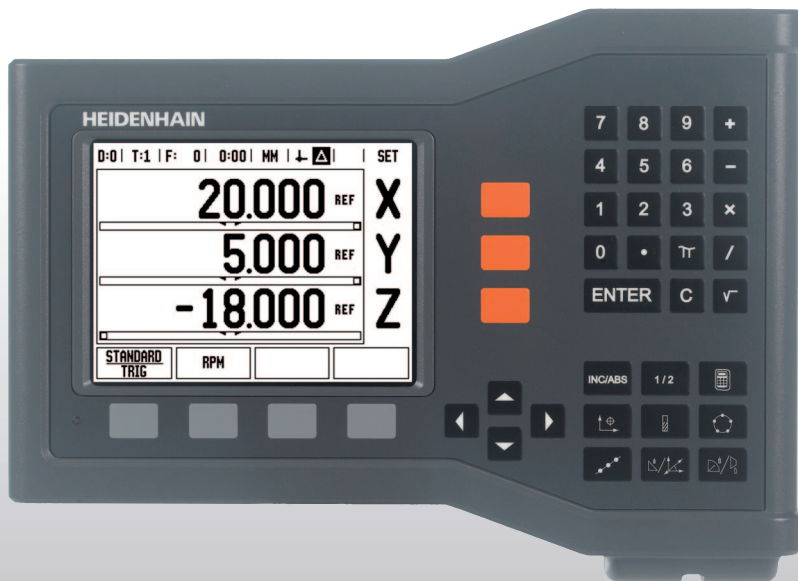




HEIDENHAIN



User's Manual

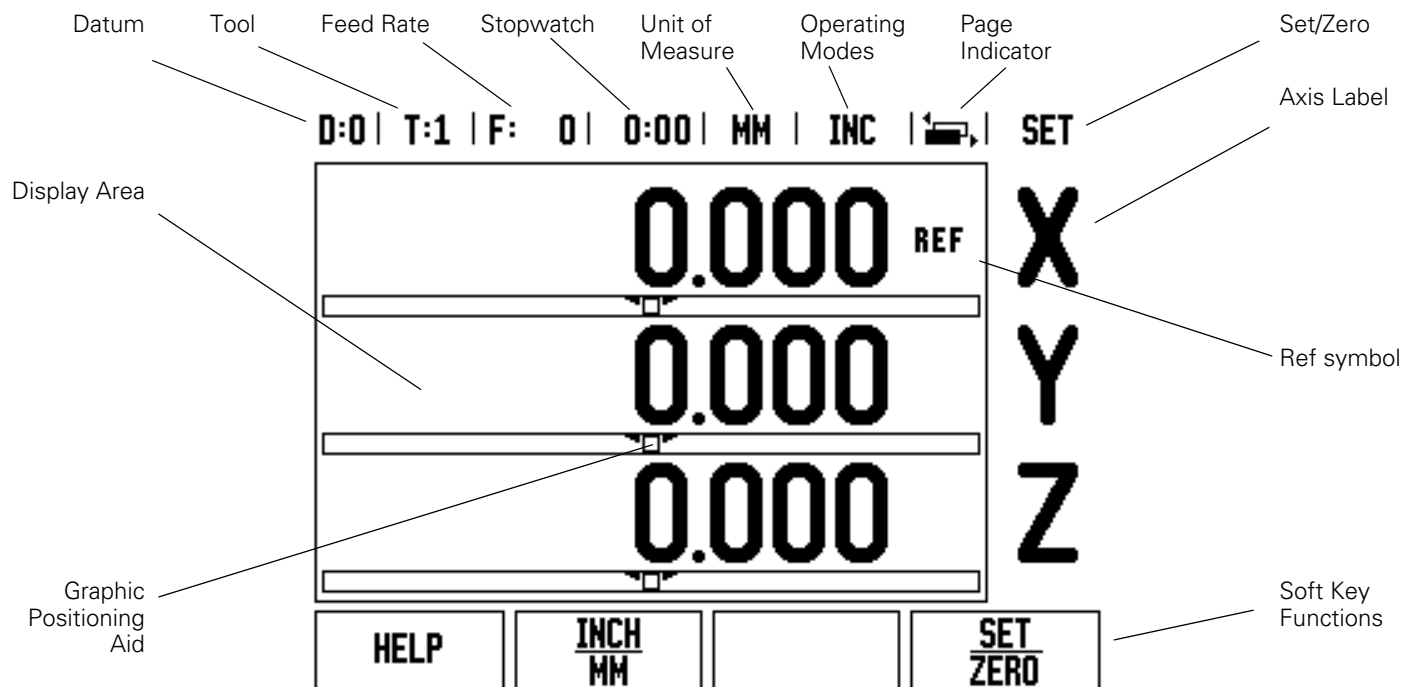
ND 522/523

English (en)
7/2007



ND 522/523 Screen

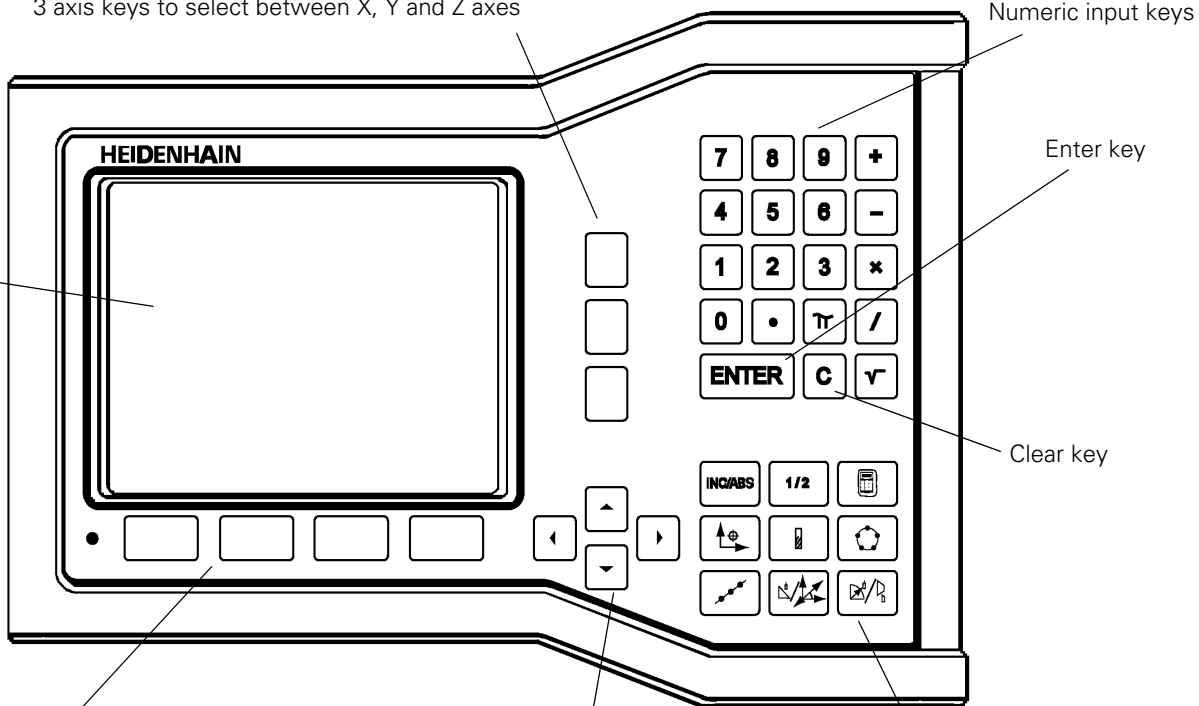
Status Bar Symbols



3 axis keys to select between X, Y and Z axes

Numeric input keys

Display Area

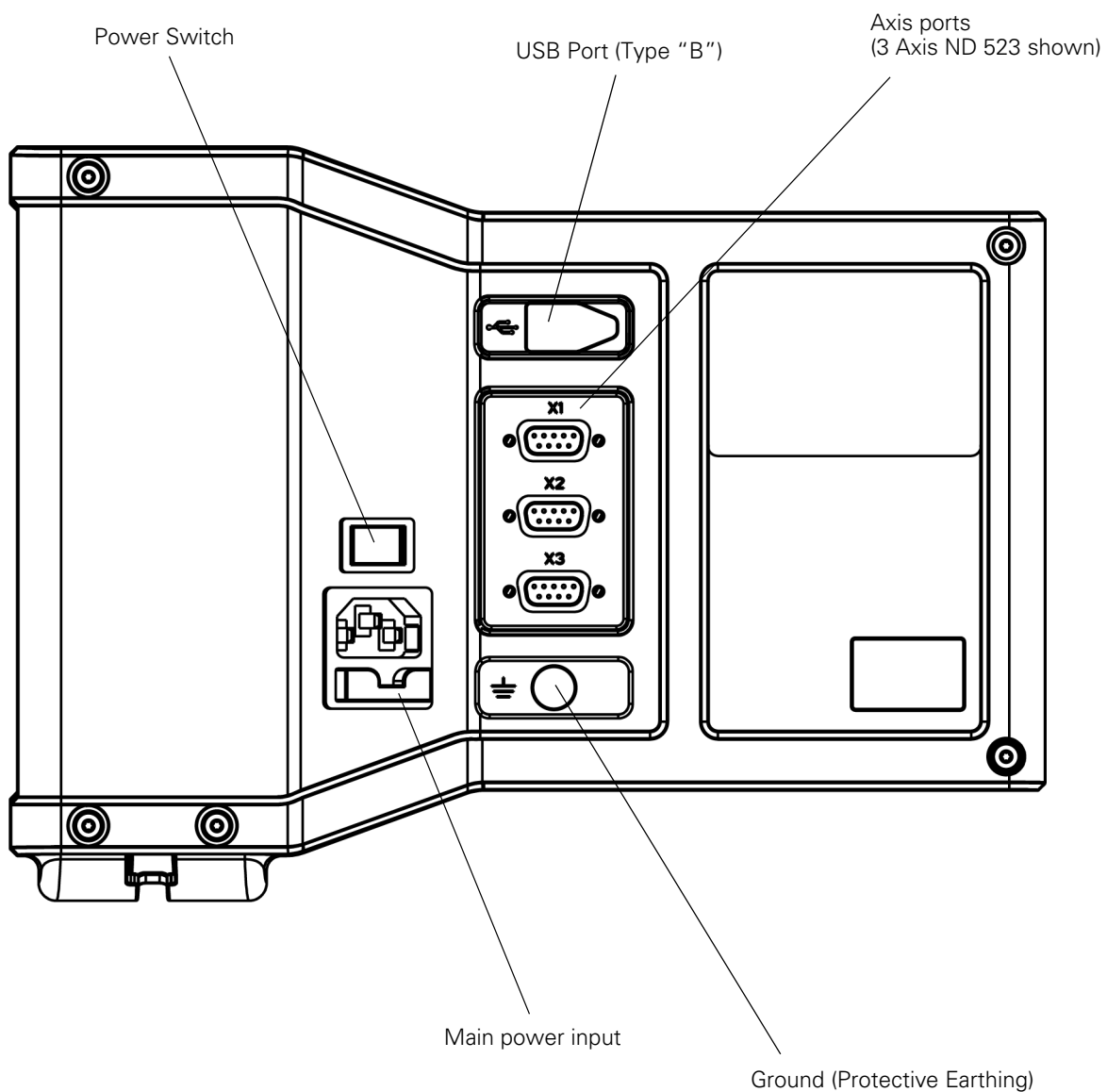


Soft keys - Row of keys under the screen of the ND 522 - 523 whose functions vary according to associated fields that appear above them on the screen

Up/Down arrow keys - are also used to adjust screen contrast



ND 522/523 Back View



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 DRO's are available in **Two axis (ND 522)** and **Three axis (ND 523)** form. The ND 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 523.



Warning – Refer to accompanying documents

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



Caution - Risk of electric shock

e.g. when opening a housing.

ND 522/523 Fonts

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

- Soft keys - **SETUP** soft key
- Hard keys - **ENTER** hard key
- Forms - *ANGULAR* forms
- Form Fields - **RADIUS**
- Data in fields - ON, OFF



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
Layout of Screen	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
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
Tool Radius Compensation feature	31
Sign for the length difference ΔL	32
Entering tool data (Fig. I.25)	32
Tool Table Usage	33
Calling the Tool Table	35
Tool call	35
Datum Hard key	35
Probing with a Tool	37
Presetting	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
Incline & Arc Milling	52
Functions for Incline and Arc Milling.	52
Incline Milling	52
Execution	53
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
Preparation:	61
Setting Datums using NOTE/SET Function	63
Taper Calculator Hard Key	64
Presetting	65
R _x (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 (see Fig. II.1)	71
Preventative maintenance	71
Connecting the Encoders	72
Pin layout for encoder inputs.	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
Keypad Test	78
Display Test	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	83
II – 6 Specifications for Milling	85
II – 7 Specifications for Turning	86
II – 8 Dimensions	87
II – 9 Accessories	88
Accessory ID Numbers	88
ND 522/523 Handle	
Id. Nr. 618025-01	88



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** while the position that the tool is to move to is called the **nominal position**. The distance from the nominal position to the actual position is called the **distance-to-go** (Incremental). See Fig. I.1

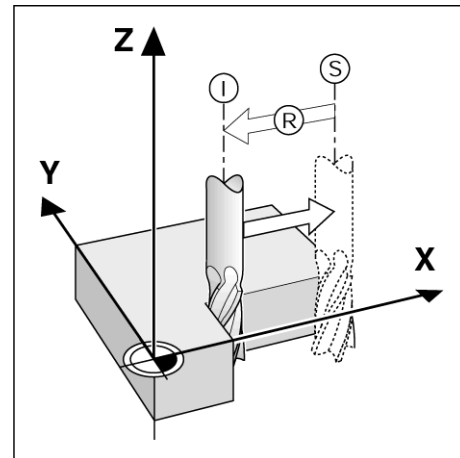


Fig. I.1 Nominal position S, actual position I and distance-to-go R

Absolute Workpiece Positions

Each position on the workpiece is uniquely identified by its absolute coordinates. See Fig. I.2.

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

X = 20 mm

Y = 10 mm

Z = 15 mm

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

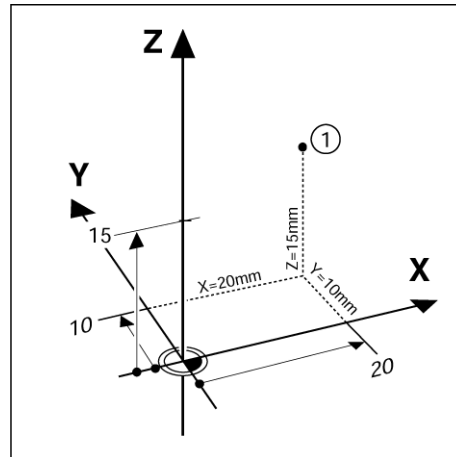


Fig. I.2 Position 1 definition through absolute coordinates

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**. See Fig. I.3

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

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

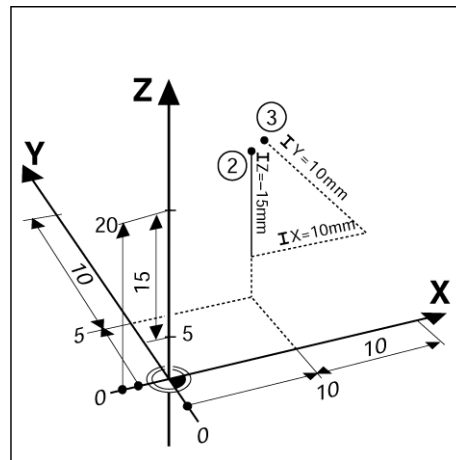


Fig. I.3 Position 3 using incremental coordinates

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. See Fig. I.4.

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

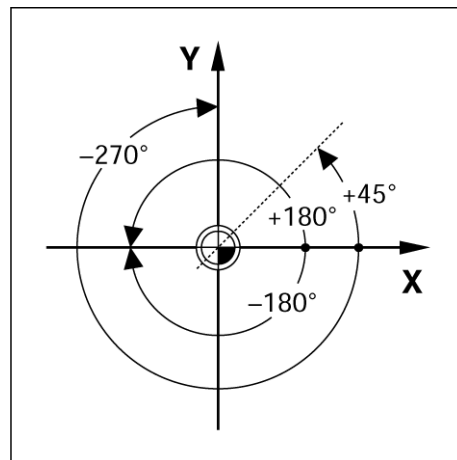


Fig. I.4 Angle and the angle reference axis, e.g. in the X / Y plane

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. See Fig. I.5.

If there is an interruption in power, the calculated position will no longer correspond to the actual position. When power is restored, you can re-establish this relationship with the aid of the reference marks on the position encoders and the ND 522/523's reference mark evaluation feature (**REF**).

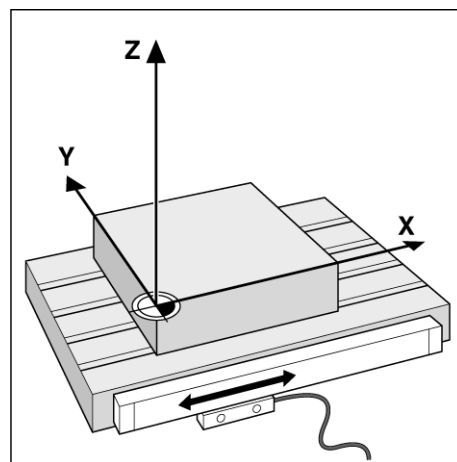


Fig. I.5 Linear position encoder, here for the X axis

Encoder Reference Marks

Encoders normally contain one or more reference marks (see Fig. I.6) 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.

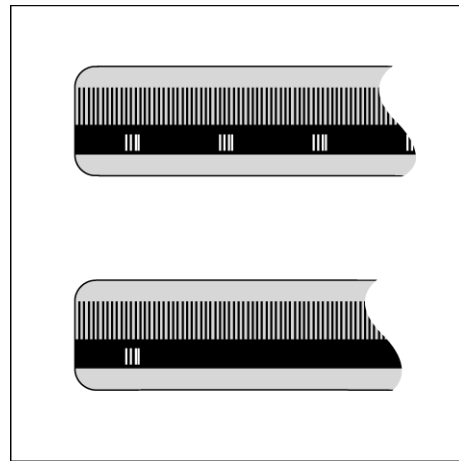


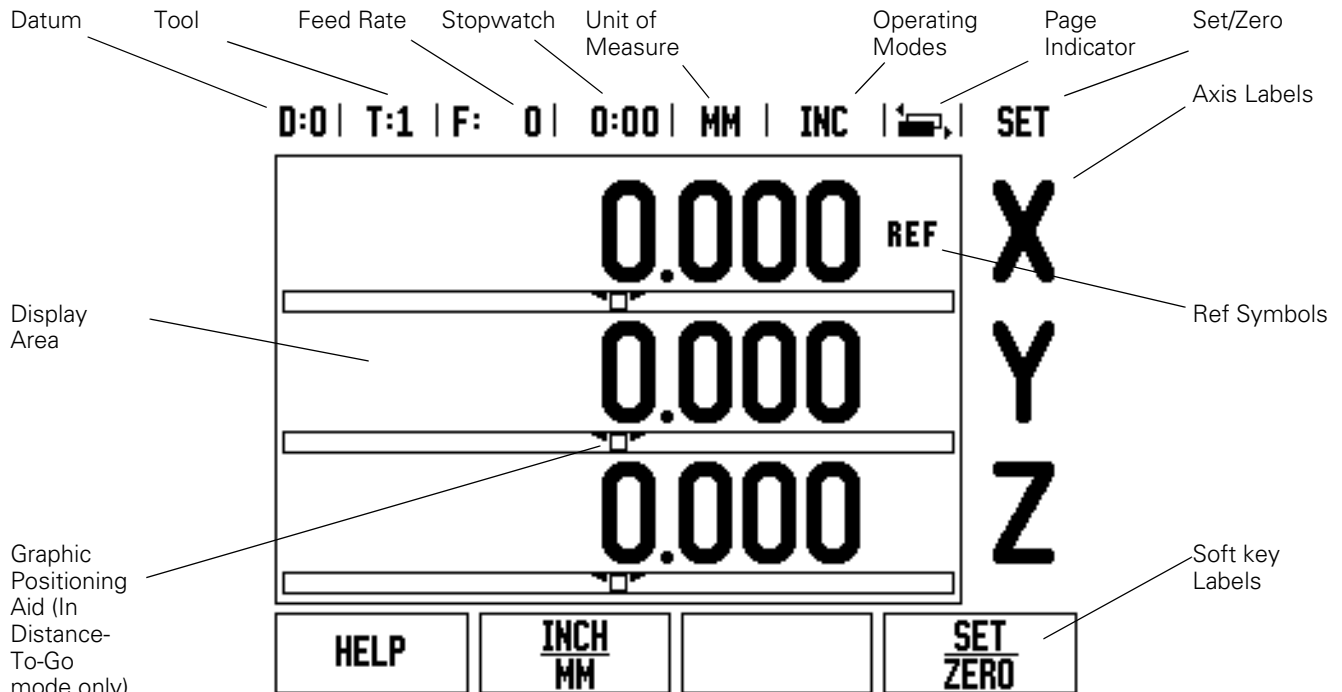
Fig. I.6 Linear scales: with distance-coded reference marks (upper illustration) and one reference mark (lower illustration)



I – 2 General Operations for ND 522/523

Layout of Screen

Status Bar Symbols












The ND 522/523 readouts provide application-specific features that allows you to obtain the most productivity from your manual machine tools.

- **Status Bar** - This displays the current datum, tool, feed rate, stop watch time, unit of measure, distance-to-go (incremental) or actual value (absolute) 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 <i>DATUM</i> form to set the datum for each axis. (Page 35)	
TOOL	Opens the <i>TOOL TABLE</i> . (Page 31 for Milling, Page 58 for Turning)	
CIRCLE PATTERN	Opens the <i>CIRCLE PATTERN</i> form. This calculates the hole positions (Page 46) for Milling	
LINEAR PATTERN	Opens the <i>LINEAR PATTERN</i> form. This calculates the hole positions (Page 49) for Milling	
INCLINE MILLING, OR VECTORING	Opens the <i>INCLINE MILLING</i> form (Page 52) for Milling, or the <i>VECTORING</i> form (Page 66) for Turning	
ARC MILLING, OR TAPER CALC	Opens the <i>ARC MILLING</i> forms (Page 55) for Milling, or the <i>TAPER CALC</i> 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 you to approach nominal positions simply by traversing to display value zero. When working within the incremental mode you can enter nominal coordinates as either incremental or absolute dimensions. The Actual Value feature (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 (See Fig. I.7) 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.



Fig. I.7 Distance-To-Go (INCREMENTAL) / Actual Value (ABSOLUTE) Hard key



Graphic Positioning Aid

When you are traversing to display value zero (in the incremental mode), ND 522/523 displays a graphic positioning aid. See Fig. I.8.

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 you want to reach.

A small square symbolizes the axis slide. An arrow indicating the direction appears in the square while the axis is moving. You can easily tell whether you are 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.

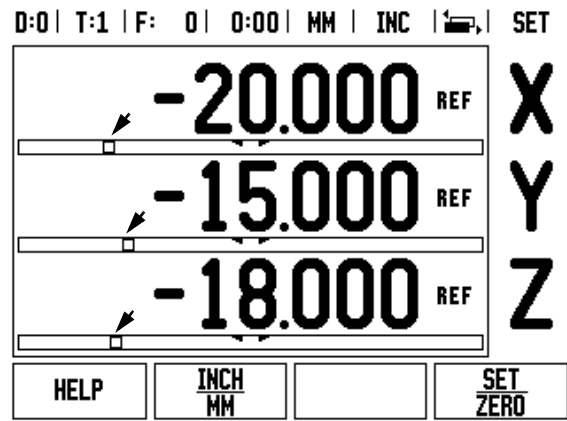


Fig. I.8 Graphic Positioning Aid in use (see arrows)

Help Screen

The integrated operating instructions provide information and assistance in any situation. See Fig. I.9

To **call** the operating instructions:

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

To view information on another topic:

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

To **leave** the operating instructions:

- ▶ Press the C key.

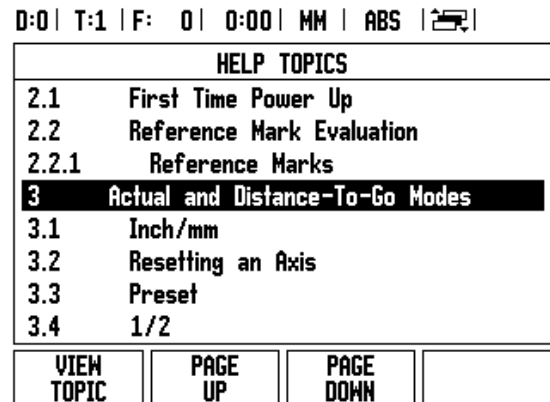


Fig. I.9 List of Topics under HELP mode




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.

You must confirm your changes by pressing the ENTER key for them to become effective. If you do not want to save your changes, press the C key to return to the previous screen without saving changes.

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. See Fig. I.10

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
ARRAY FRAME	HELP

Select the pattern type (ARRAY or FRAME).

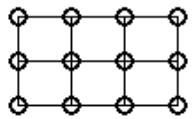


Fig. I.10 Example of Form and Instruction Box

Error Messages

If an error occurs while you are 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. See Fig. I.11 (This screen will only appear the very first time the unit is powered up. The following steps may have already been completed by the installer).

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

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

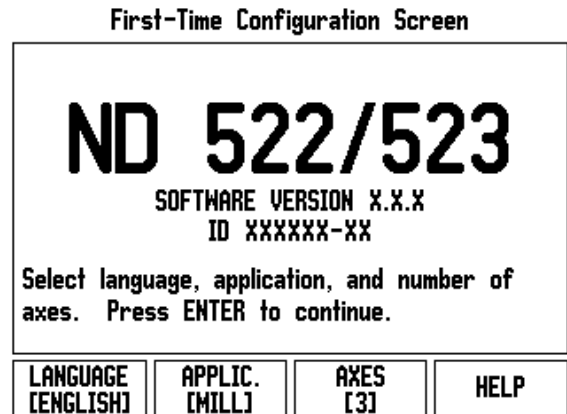


Fig. I.11 Initial screen

Your 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 you last defined by setting the datum.

If the axis encoder has reference marks, the REF indicator will flash. See Fig. I.12. After crossing over the reference marks, the indicator will stop flashing and change to non-flashing REF.

Working without reference mark evaluation

You can also use the ND 522/523 without crossing over the reference marks. Press the **NO REF** soft key to exit the reference mark evaluation routine and continue.

You can still cross over reference marks at a later time, if it becomes necessary to define 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.

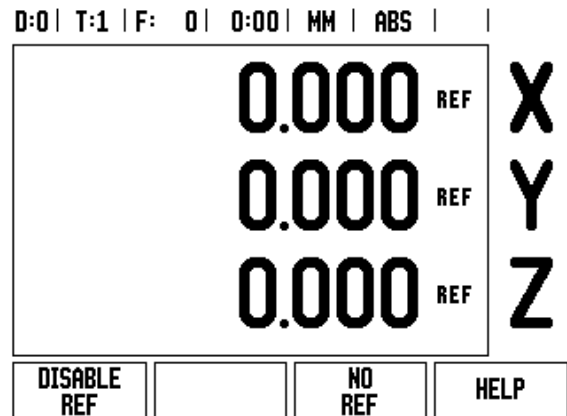


Fig. I.12 Screen for establishing Ref Marks



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. You do not have to cross over the reference marks of all the encoders, only those that you need. If all reference marks have been found the ND 522/523 will return to the DRO display screen automatically.



If you do **not** cross over the reference marks, 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. See Fig. I.13. 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.

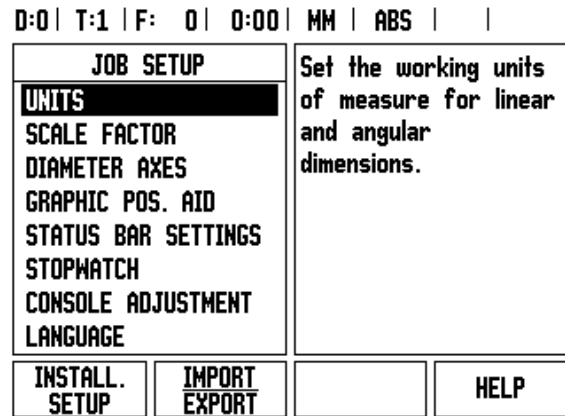


Fig. I.13 Job Setup screen in milling

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. You can also select the unit of measure 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. See Fig. I.14.

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

Mirror



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

Diameter Axes

Select Diameter Axes to set which axes can be displayed in either radius or diameter values. **ON** indicates that the axis position will be displayed as a diameter value. When **OFF**, the Radius/Diameter feature does not apply. See Fig. I.15. For turning applications see "R_x (Radius/Diameter) Soft Key" on 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 you need for that axis press **ON/OFF** soft key to turn feature on or off.
- ▶ Press ENTER.

D:0 T:1 F: 0 0:00 MM ABS			
SCALE FACTOR			
SCALE FACTOR			
X	<input checked="" type="checkbox"/>	OFF	Set a scale factor to enlarge, shrink, or mirror a part. When scale factor is ON, the ∇ indicator appears on the axis display.
Y	<input type="checkbox"/>	OFF	
Z	<input type="checkbox"/>	OFF	
ON		OFF	
		HELP	

Fig. I.14 Scale factor

D:0 T:1 F: 0 0:00 MM ABS			
DIAMETER AXES			
DIAMETER AXES			
X	<input checked="" type="checkbox"/>	ON	Set to ON to display the position as a diameter value.
Y	<input type="checkbox"/>	OFF	
Z	<input type="checkbox"/>	OFF	
ON		OFF	
		HELP	

Fig. I.15 Diameter Axes form



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. See Fig. I.16.

- ▶ 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 you want to see 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.



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

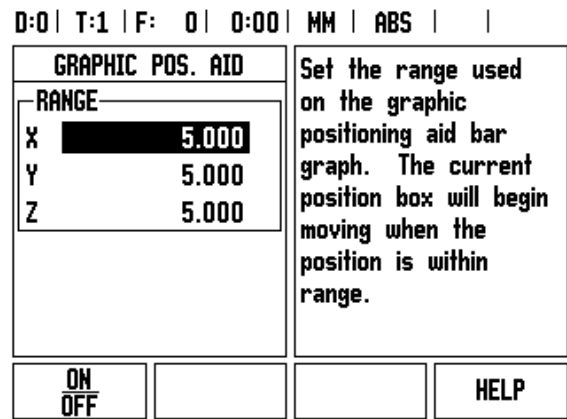


Fig. I.16 Graphic positioning aid



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 your 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 you are currently on. Each key has a reference page for added information. See Fig. I.17

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

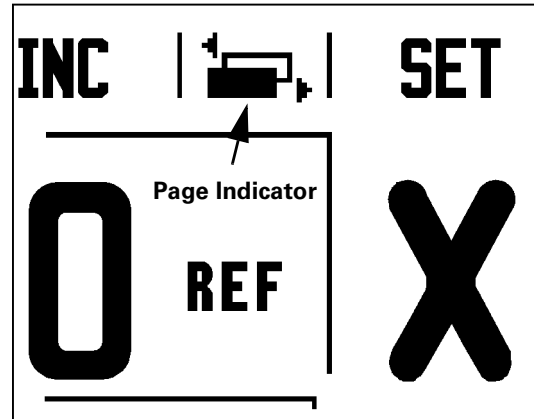


Fig. I.17 Page Indicator and Set Zero Indicator



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. See Fig. I.18

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.

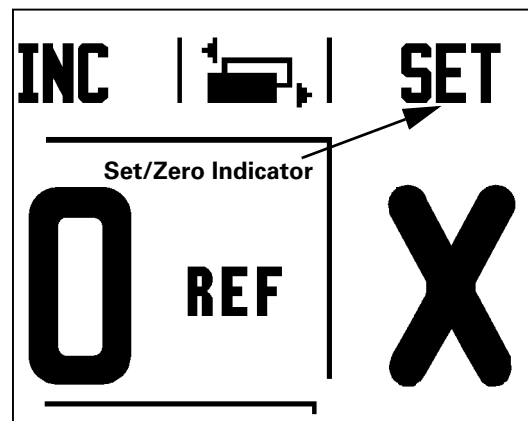


Fig. I.18 Page Indicator and Set Zero Indicator



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. See Fig. I.19



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

1/2

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

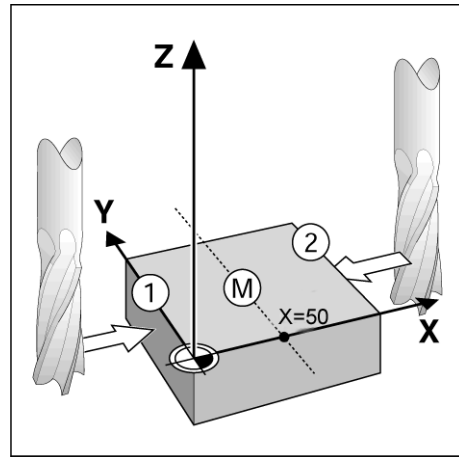



Fig. I.19 Example of finding the midpoint




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. See Fig. I.20 & Fig. I.21.

 When you need to enter more than one calculation into a numeric field, the calculator will perform multiplication and division before it performs addition and subtraction. If you were to enter $3 + 1 \div 8$, 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. Whenever you’re calculating the SIN, COS or TAN of an angle, enter the angle first and then press the appropriate soft key.

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

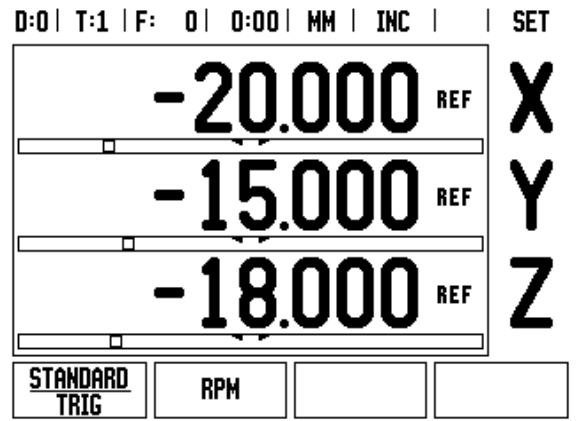


Fig. I.20 Calculator soft keys

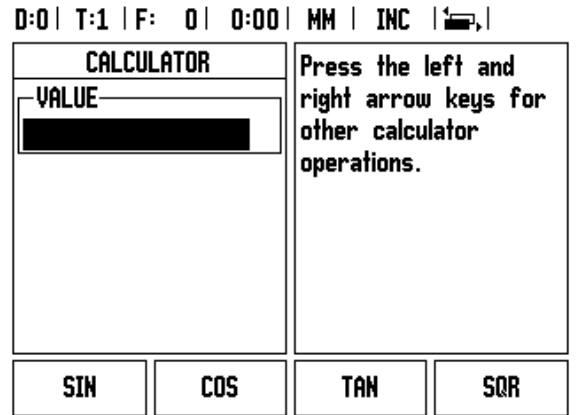


Fig. I.21 Standard/Trig function soft keys




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. See Fig. I.22

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
------------------	---------------	-------------	------

Fig. I.22 Tool Table in Milling

Tool Radius Compensation feature

ND 522/523 has a tool radius compensation feature. This allows you to enter workpiece dimensions directly from the drawing. The displayed Incremental is then automatically lengthened (R+) or shortened (R-) by the value of the tool radius. See Fig. I.23. (For more information see "Presetting" on page 39)

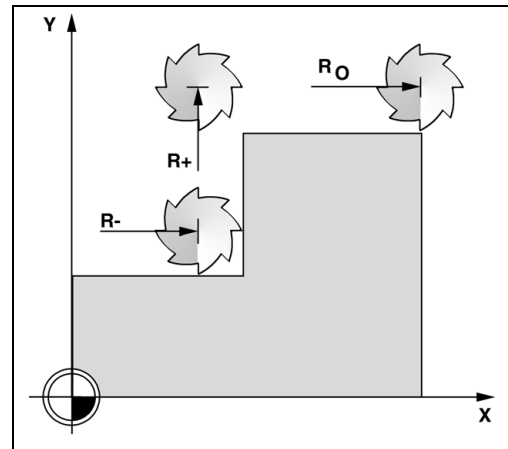


Fig. I.23 Tool radius compensation



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 <i>TOOL TYPES</i> 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 in Fig. I.24.

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

Entering tool data (Fig. I.25)

- ▶ Choose T00L 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.

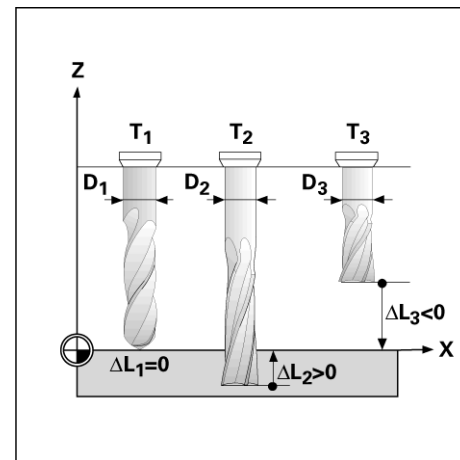




Fig. I.24 Tool length and diameter

Tool Table Usage


Example: Setting a workpiece datum without using the probing function. See Fig. I.25

- 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 T00L hard key.
The cursor will default to the *TOOL TABLE* form.

TOOL TABLE



 Cursor to the tool you want to define or enter the tool number. Press ENTER.

TOOL DIAMETER

 Enter the tool diameter (2) cursor down to the LENGTH field.

 Press the DOWN ARROW key.

TOOL LENGTH

  Enter the tool length for example, (20 mm) and cursor down to the UNITS field.



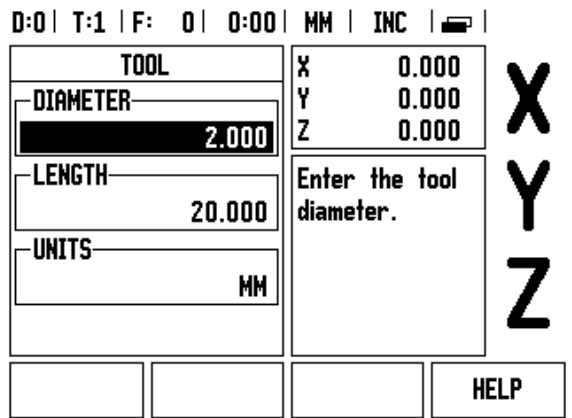


Fig. I.25 Tool length and diameter entry



- 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, you will not be able to switch between the new tools and the existing tools without having to re-establish the datum. Before adding the new tools, select one of the tools from the tool table. Touch the tool to a reference surface and set the datum to 0.

TOOL UNIT

INCH
MM

▶ Enter the TOOL UNIT (inch/mm) and



▶ cursor to the TOOL TYPE field.

TOOL TYPE

TOOL TYPES

▶ Press **TOOL TYPES** soft key. Select from list of tools and press ENTER. (See Fig. I.26)

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

Fig. I.26 TOOL TYPE field

Calling the Tool Table

Before you start machining, select the tool you are using from the tool table. ND 522/523 then takes into account the stored tool data when you work with tool compensation.

Tool call



Press the T00L hard key.

TOOL NUMBER



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



Verify the proper tool has been called, and press the 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 you probe the workpiece with an edge of a tool.

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

The datum table can hold up to 10 datum points. In most cases this will free you 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. See Fig. I.27 & Fig. I.28.

Tool diameter: $D = 3\text{ mm}$

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

- 1 . 5

Enter the position of the tool center ($X = -1.5\text{ mm}$) and



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



Touch the workpiece at edge **2**.

DATUM SETTING Y

- 1 . 5

Enter the position of the tool center ($Y = -1.5\text{ mm}$) and



press the **DOWN ARROW** key.



Touch the workpiece surface.

DATUM SETTING Z = + 0

0

Enter the position of the tool tip ($Z = 0\text{ mm}$) 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		Y	0.000
1		Z	0.000
DATUM		Enter the new actual position of the tool or press PROBE.	
X	-1.500	X Y Z	
Y	-1.500		
Z	0		
PROBE			HELP

Fig. I.27 SET DATUM form

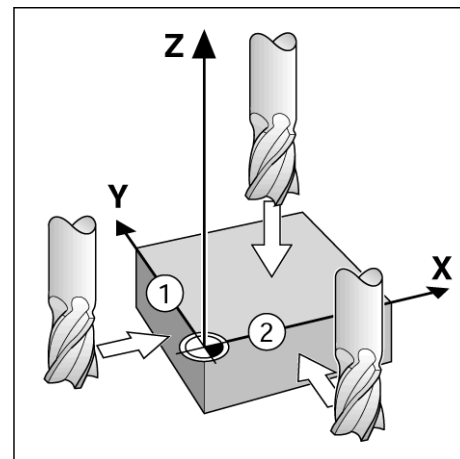


Fig. I.28



Probing with a Tool

Using a tool to set datum points using the ND 522/523's probing functions. See Fig. I.29 & Fig. I.30.
 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

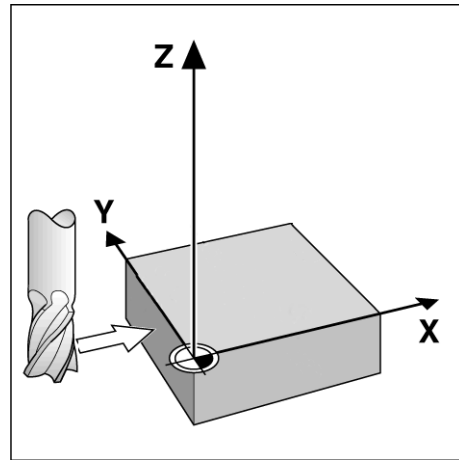






Fig. I.29 Setting datum using an edge

 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.		
X				
Y				
Z				
EDGE		CENTER LINE	CIRCLE CENTER	HELP

Y
Z

Fig. I.30



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. See Fig. I.31

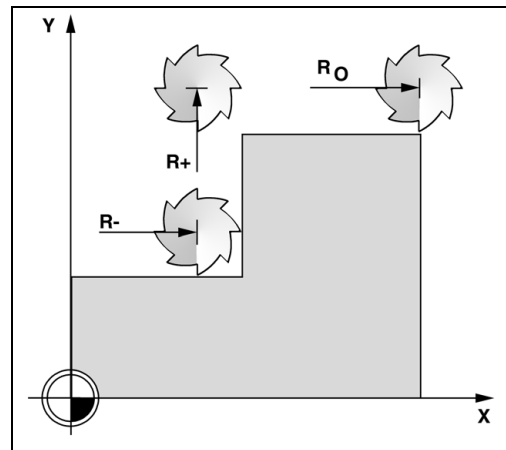



Fig. I.31 Tool radius compensation

Absolute Distance Preset

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

The coordinates are entered as absolute dimensions; the datum is the workpiece zero. See Fig. I.32 & Fig. I.33.

- 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

 If you would like to recall the last entered preset for a particular axis, press the axis key.

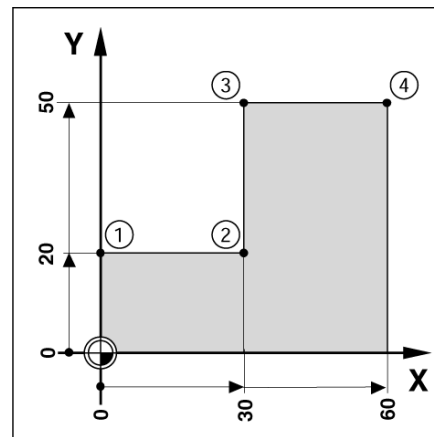


Fig. I.32 Single cycle preset



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 so that you are in Set mode.

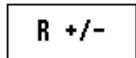


Press the Y axis key.

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

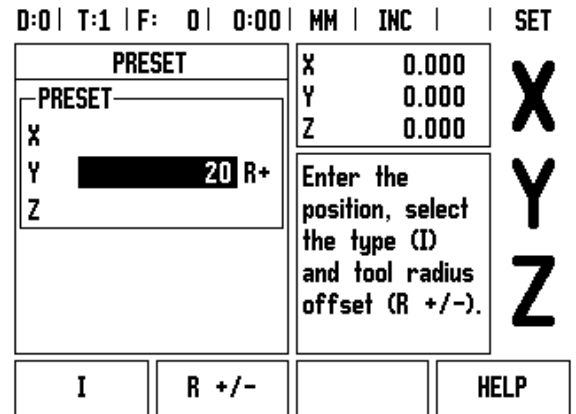


Fig. I.33

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. See Fig. I.34 & Fig. I.35.

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

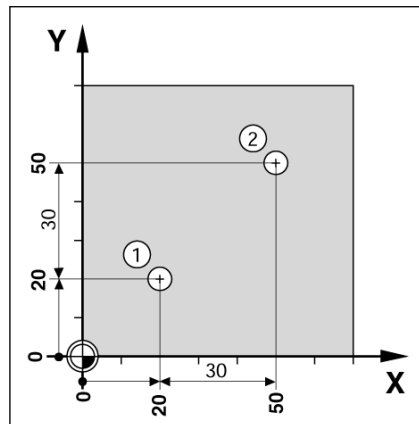


Fig. I.34 Drilling example



Press the **SET/ZERO** soft key so that you are in Set mode.



Press the X axis key.

- NOMINAL POSITION VALUE -

2 0

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

2 0

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



NOMINAL POSITION VALUE

- 1 2

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 your 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 your 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.

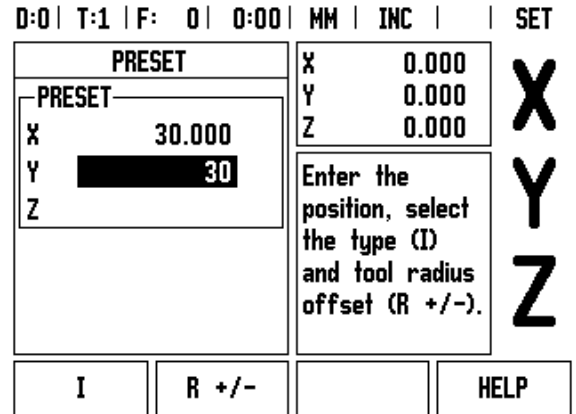


Fig. I.35 Drilling example

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. See Fig. I.36. The values shown in this Figure are only an example. Consult your tool manufacturer’s manual to verify spindle speed ranges per tool.

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

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

- ▶ Press the **UNITS** soft key to show the units as inch or millimeter.
- ▶ The *RPM CALCULATOR* form is closed by pressing the **C** key.

D:0 T:1 F: 0 0:00 MM INC			
RPM CALCULATOR		Enter the diameter of the rotating tool or part. The spindle speed will be calculated.	
DIAMETER			
5.0000 MM			
SURFACE SPEED			
47.1239 M/MIN			
SPINDLE SPEED			
3000.0 RPM			
UNITS			HELP

Fig. I.36 RPM Calculator form



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 you start machining. It is also useful when: selecting holes directly, executing holes separately, and skipping holes.

Functions for milling patterns

Function	Soft key
Press this to see the layout of the current pattern.	VIEW
Press to go to previous hole.	PREVIOUS HOLE
Press to manually advance to the next hole.	NEXT HOLE
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 "Zero Angle Reference Axis" on 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 you then move to simply by traversing to display value zero.



Example: Enter data and execute a circle pattern. (See Fig. I.37, Fig. I.38 & Fig. I.39.)

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



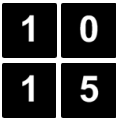
Enter the type of circle pattern (full). Cursor to the next field.

HOLES



Enter the number of holes (4).

CIRCLE CENTER



Enter the X and Y coordinates of the circle center (X = 10), (Y = 15). 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").

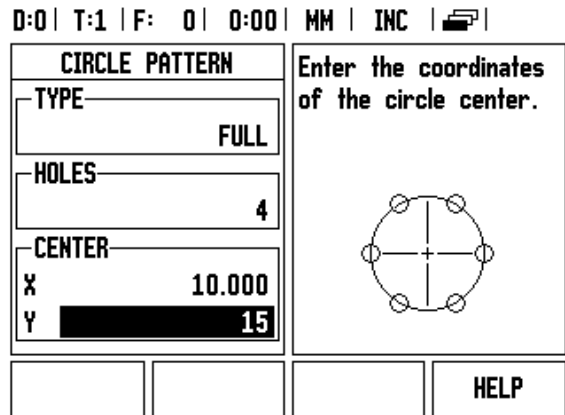


Fig. I.37 Beginning of Circle Pattern form

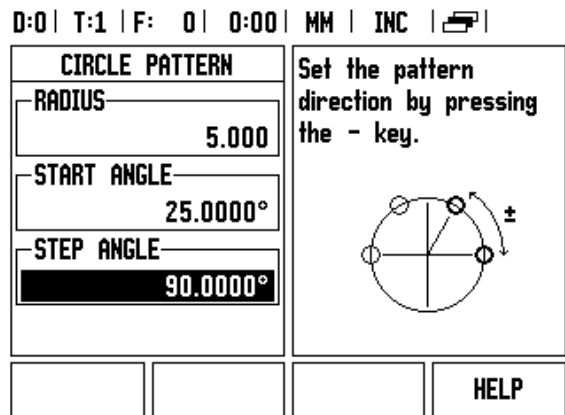


Fig. I.38 Page 2 of Circle Pattern Form

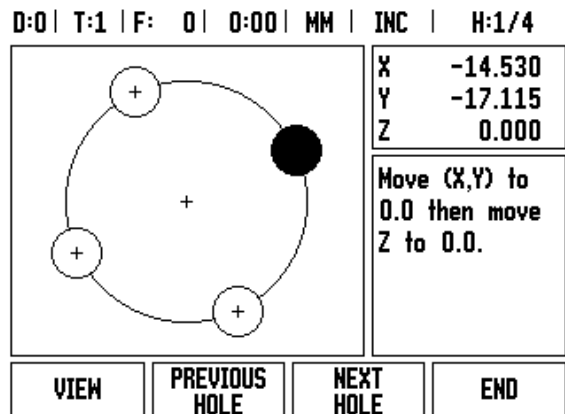


Fig. I.39 View of Circle Pattern Graphic



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. (See Fig. I.40, Fig. I.41 & Fig. I.42).

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°).

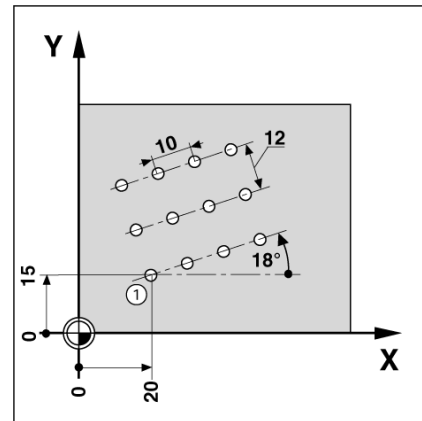


Fig. I.40 Linear Pattern Example

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		<div style="display: flex; justify-content: space-between;"> ARRAY FRAME HELP </div>
4		

Fig. I.41 Linear Pattern Form

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

Fig. I.42 View of Linear Pattern Graphic



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.



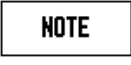




Incline & Arc Milling

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 to execute the milling operation.	
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: See Fig. I.43, and Fig. I.44.

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



Execution

- Execute the milling operation by opening the incline milling form and pressing the run soft key or enter key. The screen switches to the incremental DRO view.
- Initially, the DRO shows the current incremental moving distance from the start point. Move to the start point and make a plunge cut or the first pass across the surface. Press the Next pass soft key to continue with the next step along the contour.
- After pressing **NEXT PASS**, the incremental display shows the distance from the next step along the line's contour.
- If no step size was specified, the incremental display always shows the distance from the closest point on the line. To follow the contour, move the two axes in small steps, keeping the (X, Y) positions as close to 0 as possible.
- 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:
(See Fig. I.43).

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

Fig. I.43 Entry Form: Plane

D:0 T:1 F: 0 0:00 MM ABS			
MILL LINE			
END POINT			
X	0.000		
Y	0.000		
STEP			
0.000			
NOTE	<table border="1"> <tr> <td>Enter the END point.</td> </tr> </table>	Enter the END point.	
Enter the END point.			
	<table border="1"> <tr> <td>RUN</td> <td>HELP</td> </tr> </table>	RUN	HELP
RUN	HELP		

Fig. I.44 Entry Form: End Point



Arc Milling

Entry Form: See Fig. I.45, Fig. I.46, and Fig. I.47

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 too 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:
 (See Fig. I.45), (Fig. I.46), and (Fig. I.47).

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.

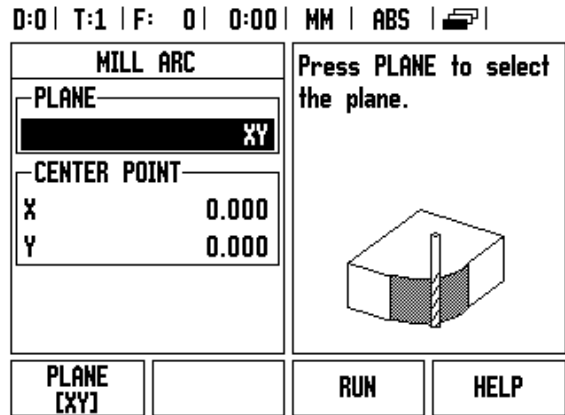


Fig. I.45 Entry Form: Arc

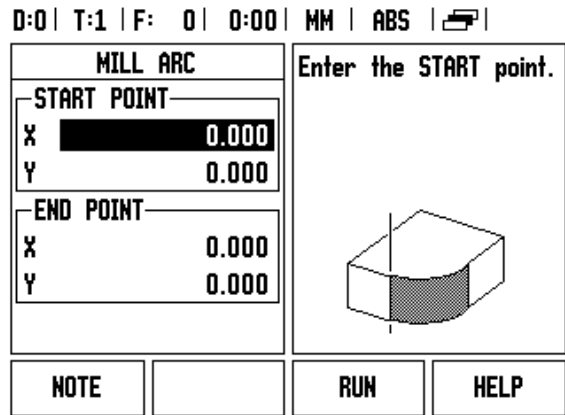


Fig. I.46 Entry Form: Start Point

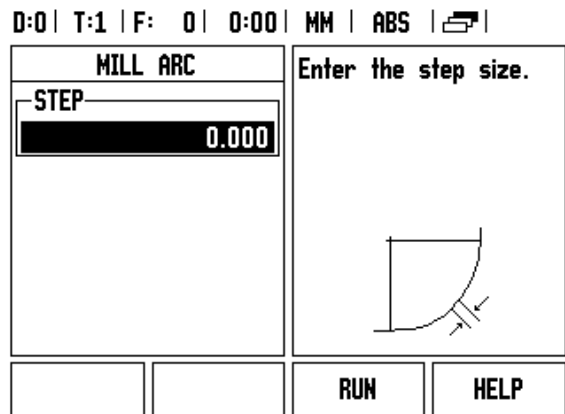


Fig. I.47 Entry Form: End Point



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 you change a workpiece and establish a new datum, all tools are automatically referenced from the new datum.

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

If you have measured your tools using a tool presetter, the offsets may be entered directly. See Fig. I.48.

To access the Tool Table menu:




Press the T00L hard key.

The cursor will default to the TOOL TABLE field.

TOOL TABLE



Scroll to the tool you want to define. 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	

Fig. I.48 Tool table in turning



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. See Fig. 1.49

Touch the known diameter in the X axis.



Press the T00L 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) if you 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.

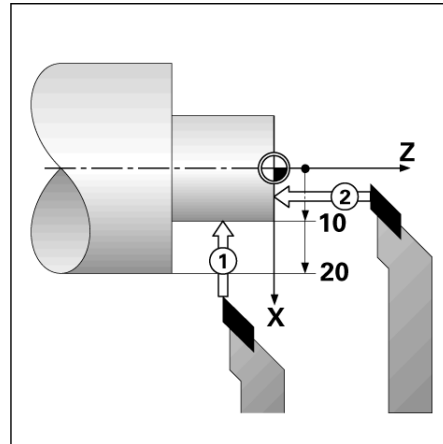


Fig. 1.49



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. See Fig. I.50.

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 **T00L** hard key. Select the desired tool and press **ENTER** key.



Select the **X** axis key.

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 or radius, for example, 15 mm and press **ENTER**.

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

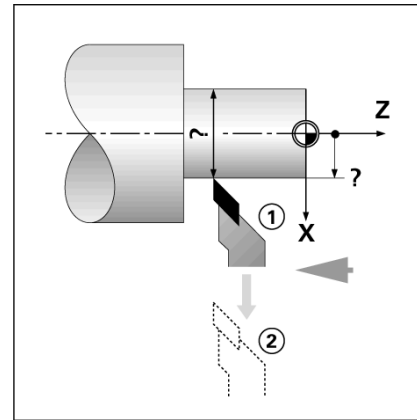


Fig. I.50 Setting Tool Offset

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

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

Fig. I.51 TOOL/SET Form



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. See Fig. I.52 & Fig. I.53.

Axis sequence in this example: X - Z

Preparation:

Call the tool data by selecting the tool which you are using to touch the workpiece.



Press the DATUM hard key.



The cursor will be in the DATUM NUMBER field.



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



Touch the workpiece at point **1**.

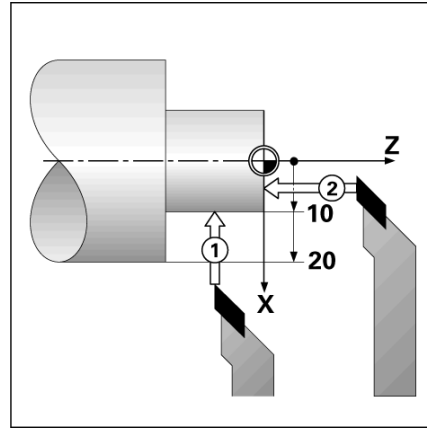


Fig. I.52 Setting a workpiece datum

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

SET DATUM		X	0.000 \emptyset	X
DATUM NUMBER		Z	0.000	
2		Face part in Z then press NOTE or enter tool position.		Z
DATUM		X	20.000 \emptyset	
Z		Z	0.0	
NOTE			HELP	

Fig. I.53



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 (Ø) if you input a diameter value.

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



Touch the workpiece surface at point **2**.

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. See Fig. I.54 & Fig. I.55

To use the NOTE/SET function:



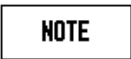
Press the DATUM hard key.

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.



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, for example, 15 mm and press ENTER.

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

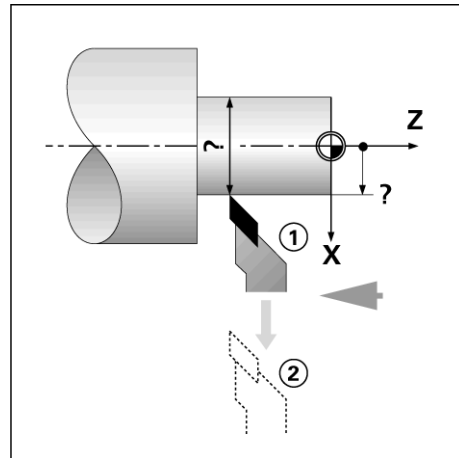


Fig. I.54

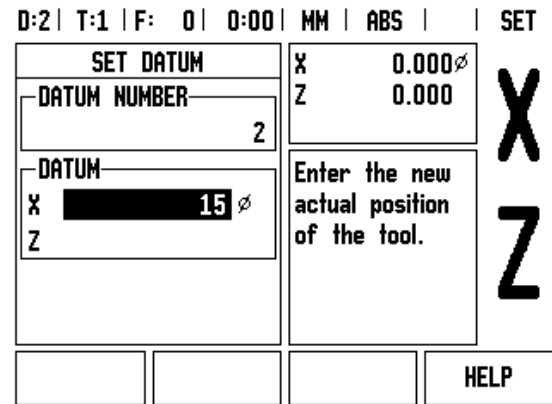


Fig. I.55 Setting Datum using NOTE/SET



Taper Calculator Hard Key

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

Use the taper calculator to calculate taper angle. See Fig. I.56 and Fig. I.57.

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.

You will notice the soft key selection has changed and now includes the taper calculator functions.

D1/D2 LENGTH



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



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			
	25.0000		
ANGLE			
	5.7106°		
NOTE			HELP

Fig. I.56 Taper Calculator form - Diameter 1

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

TAPER CALCULATOR		Enter first diameter.	
DIAMETER			
D1	10.0000		
D2	5.0000		
LENGTH			
	25.0000		
ANGLE			
	5.7106°		
NOTE			HELP

Fig. I.57 Taper Calculator form - Diameter 2



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 you are entering 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 **R_X** soft key (see below).

R_X (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. See Fig. I.58.

Example: Radius display, position 1 X = 20 mm

Diameter display, position 1 X = \varnothing 40 mm

R_X

Press the **R_X** soft key to switch between radius display and diameter display.

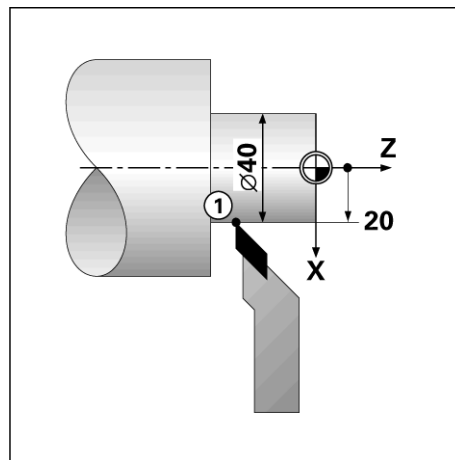


Fig. I.58 Workpiece for radius/diameter display

Vectoring Hard Key

Vectoring breaks down the movement of the compound axis into the crossfeed or longitudinal axes. See Fig. I.59. If you are turning threads, for example, vectoring lets you see the diameter of the thread in the X-axis display, even though you are moving the cutting tool with the compound axis handwheel. With vectoring enabled, you can preset the desired radius or diameter in the X-axis, so that you can “machine to zero”.



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

Press the **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**.

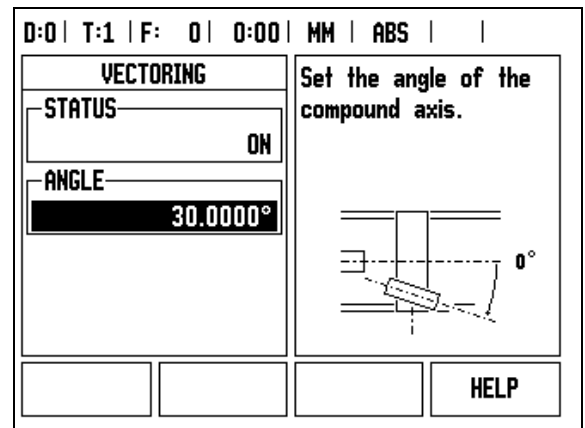


Fig. I.59 Vectoring

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. See Fig. I.60.

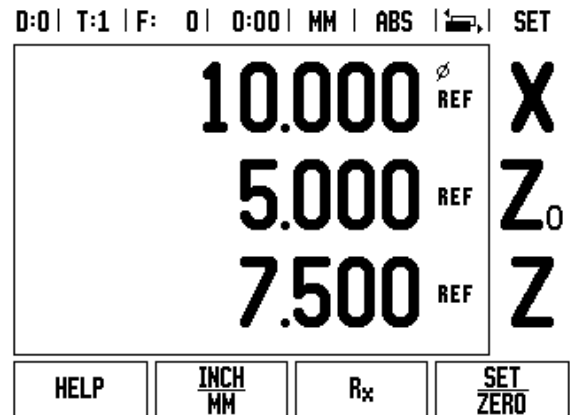


Fig. I.60 Normal Display form

Enabling Z Coupling

To couple the Z₀ and Z axis and have the result displayed on the Z₀ display, press and hold the Z₀ key approximately 2 seconds. The sum of the Z positions will be displayed on the Z₀ display and the Z display will be blanked. See Fig. I.61.

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.

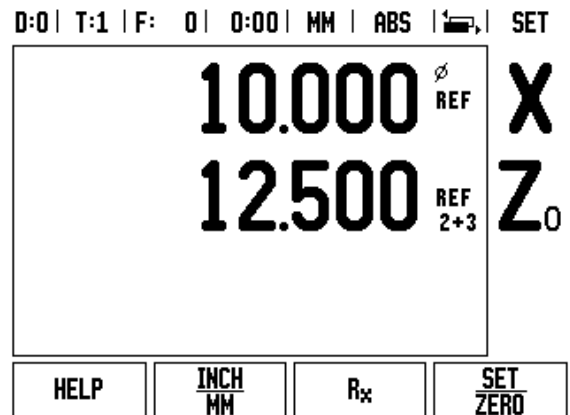


Fig. I.61 Enabling Z Coupling



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. Nr. 618025-01" on page 88.

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	100 - 240 Vac
Power	25 VA max.
Frequency	50/60 Hz
Fuse	500 mA/250 Vac, 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 (see Fig. II.1)

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, see Fig. II.2.

Preventative maintenance

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

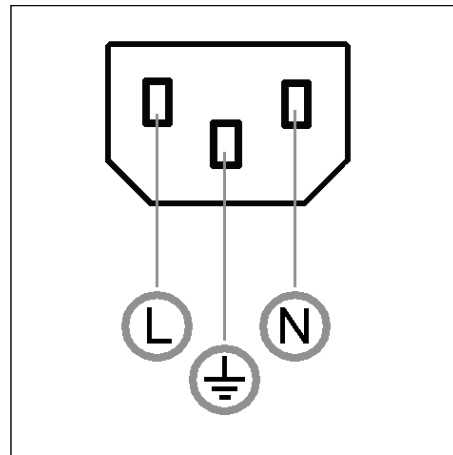


Fig. II.1 Power connector

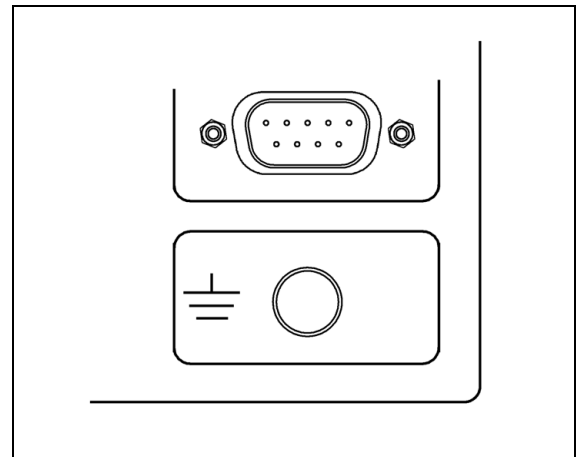


Fig. II.2 The protective ground terminal on the rear panel.



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	N/C
2	Ua 1
3	$\overline{Ua 1}$
4	Ua 2
5	$\overline{Ua 2}$
6	0 V (Un)
7	+ V (Up)
8	$\overline{Ua 0}$
9	Ua 0

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

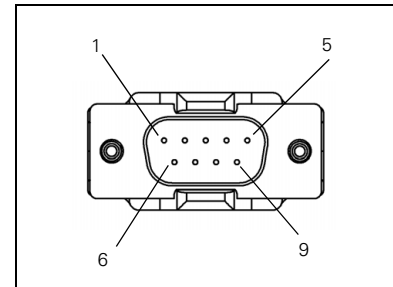


Fig. II.3 9 pin header X1 - X3 plug pinout for encoder signal input on the back side of the ND 522/523.

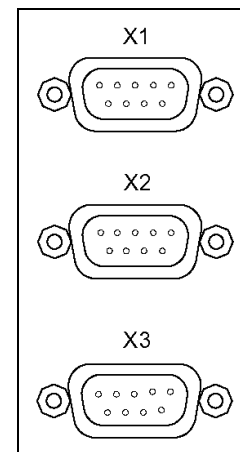


Fig. II.4 Encoder inputs on the back side of the ND 522/523.



II – 2 Installation Setup

Installation Setup Parameters

Installation setup is accessed by pressing the **SETUP** soft key, which brings up **INSTALLATION SETUP** soft key. See Fig. II.5

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		Set the counter application (MILL or TURN) and number of axes.	
ENCODER SETUP			
DISPLAY CFG.			
ERROR COMPENSATION			
BACKLASH COMP.			
COUNTER SETTINGS			
DIAGNOSTICS			
JOB SETUP		HELP	
IMPORT EXPORT			

Fig. II.5 Installation screen

Encoder Setup

The ENCODER SETUP is used to set the encoder resolution and type (linear, rotary), count direction, reference mark type. See Fig. II.6.

- ▶ The cursor will default to the **ENCODER SETUP** field upon opening Installation Setup. Press **ENTER**. This opens a list of possible encoder inputs.
- ▶ Scroll to the encoder you want to change and press ENTER.
- ▶ Cursor will be in the ENCODER TYPE field, select the encoder type by pressing the **LINEAR/ROTARY** soft key.
- ▶ For linear encoders, cursor to the RESOLUTION field and use **COARSER** or **FINER** soft keys to select the encoder's resolution in μm (10, 5, 2, 1, 0.5) or type in the exact resolution. For rotary encoders, enter the number of lines per revolution.
- ▶ In the REFERENCE MARK field, toggling the **REF MARK** soft key to select whether the encoder has no reference signal with **NONE**, single reference mark with the **SINGLE** or with the **CODED** soft key for encoders with the distance coded reference.
- ▶ 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)		Select the encoder type (LINEAR or ROTARY).	
ENCODER TYPE			
LINEAR			
RESOLUTION			
5.0 μm			
REFERENCE MARK			
CODED			
LINEAR		HELP	
ROTARY			

Fig. II.6 Encoder Setup form



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 (i.e. "2 + 3". See Fig. II.8.

Error Compensation

The distance a cutting tool travels, measured by an encoder, can in certain cases, differ from the actual tool travel. This error can occur due to ball screw pitch error or deflection and tilting of axes. This error can either be linear or non-linear. You can determine these errors with a reference measurement system, 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. See Fig. II.7 & Fig. II.8

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

To calculate the linear error compensation use this formula:

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

with S = measured length with reference standard
M = measured length with device at axis

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

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

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

Fig. II.7 Linear error comp., calculation formula

Non-Linear Error Compensation

Non-linear error compensation should be applied, if the results of the comparison with a reference standard show an alternating or oscillating deviation. The required correction values are calculated and entered in a table. 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, you can move to the location of the start point and press **TEACH POSITION**. Press ENTER.

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

ERROR COMPENSATION		Error compensation for this input is OFF. Press TYPE to select linear or non-linear error compensation.	
INPUT X1	0 PPM		
INPUT X2	OFF		
INPUT X3	OFF		
TYPE [OFF]			HELP

Fig. II.8 Linear error compensation form



Configuring the Compensation Table

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

Reading the Graph

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

Viewing the Compensation Table

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

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

Exporting the Current Compensation Table

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

Importing a New Compensation Table

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



Backlash Compensation

When using a rotary encoder with a lead screw, a change in direction of the table might cause an error in the displayed position due to clearances within the lead screw assembly. This clearance is referred to as backlash. This error can be compensated for by inputting the amount of backlash within the lead screw into the Backlash Compensation feature. See Fig. II.9.

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.	
INPUT X1	0.2
INPUT X2	OFF
INPUT X3	OFF
Specify the amount of backlash between the encoder and the machine.	
ON OFF	HELP

Fig. II.9 Backlash compensation form



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. See Fig. II.10

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			
APPLICATION		MILL	
NUMBER OF AXES		3	
POSITION RECALL		OFF	
MILL TURN		FACTORY DEFAULTS	
		HELP	

Set the application to MILL or TURN.

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

Fig. II.10 COUNTER SETTINGS form

Diagnostics

The *DIAGNOSTICS* form provides access for testing the keypad, and display. See Fig. II.11

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.	

Fig. II.11 DIAGNOSTICS form



II – 3 Encoder Parameters

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

Example settings for HEIDENHAIN linear encoders

Encoder	Resolution	Reference marks
LS 328C LS 628C	5µm	Single/1000
LB 382 with IBV 101	1µm	Single
LB 382C with IBV 101	1µm	Single/2000
LS 378C LS 678C	1µm	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 what you need to know 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 your Heidenhain distributor for further details.) This software processes the data being sent or received over the serial cable link. All data transferred between ND 780 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

You can operate the display unit 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’
----------------	---------

<ESC>T9006<CR>	Key ‘6’
----------------	---------

<ESC>T9007<CR>	Key ‘7’
----------------	---------



Sequence of commands	Function
<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'/'Z ₀ '
<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'
<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

If you have a PC you can 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 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 <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	100 V to 240 V; 50 Hz to 60 Hz; power consumption 25 VA max.
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

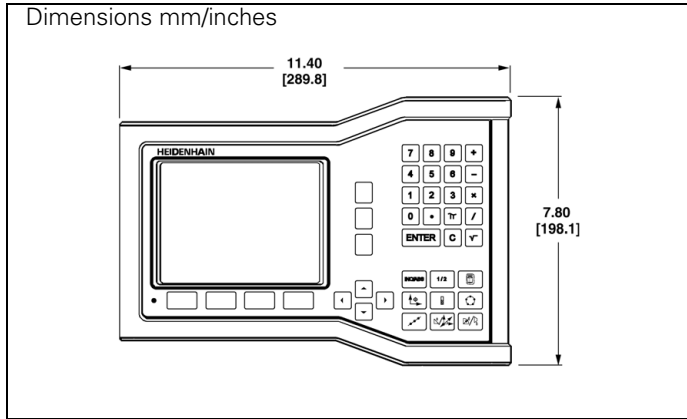


II – 7 Specifications for Turning

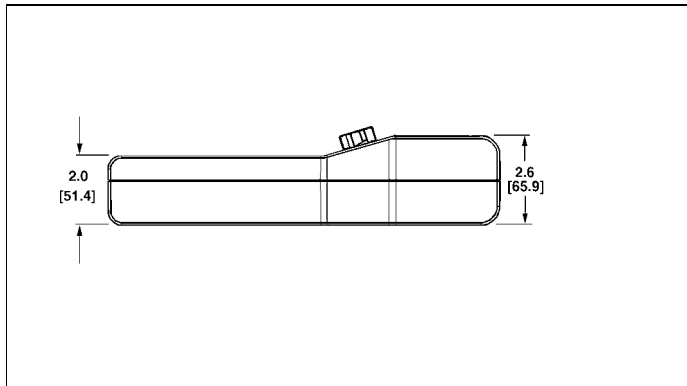
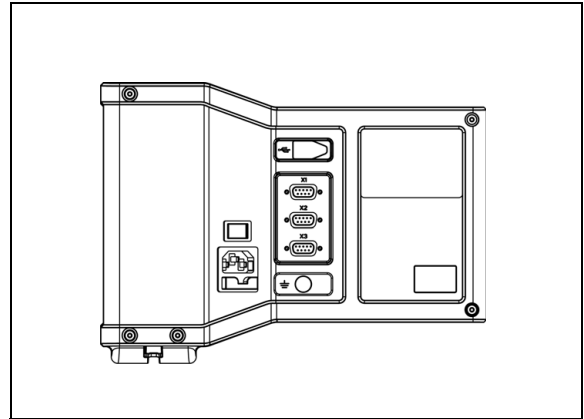
ND 522/523 Data	
Axes	2 or 3 axes from A to Z, Z ₀ , 0-9
Encoder inputs	Digital TTL Signal levels; input frequency max. 100 kHz for incremental HEIDENHAIN encoders
Display step	Linear axes: 1 mm to 0.1 µm Rotary axes: 1° to 0.0001° (00°00'01")
Display	Monochrome display for position values, dialogue and input display, graphic functions, graphic positioning aid ■ Status display: tool number, operating mode, REF, inch/mm, scaling factor, feed rate, diameter display Ø, 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	100 V to 240 V; 50 Hz to 60 Hz; power consumption 25 VA max.
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

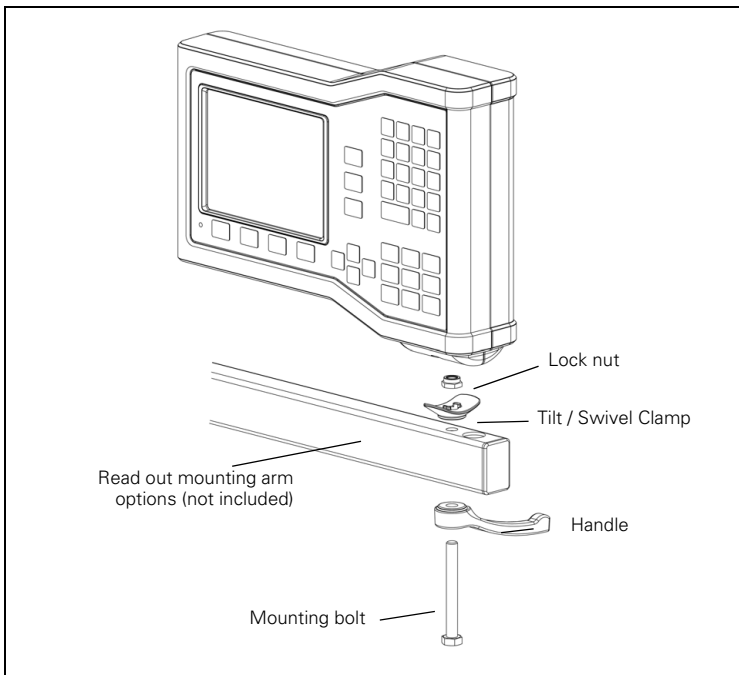
Accessory ID Numbers

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

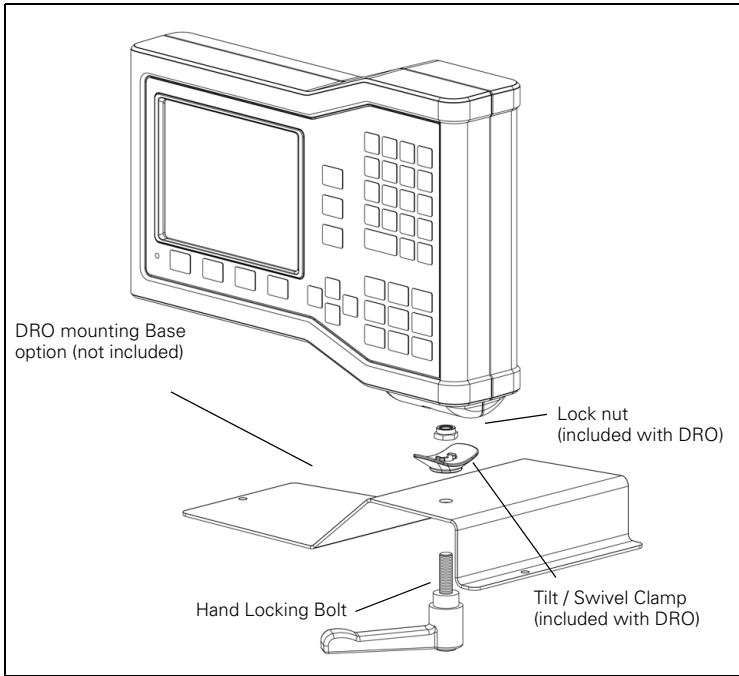
Id. Nr. 618025-01

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 ... 88
 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 ... 87
 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 ... 88
 Help Screen ... 19

I

Import/Export (setting) ... 26
 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
 Layout of Screen ... 16
 Left/Right arrow keys ... 18
 Linear Error Compensation ... 75
 Linear Pattern ... 49, 52

M

Measured Value Output ... 83
 Milling Specific Operations and Soft Key Functions Detailed ... 31
 Mirror ... 24
 Mounting Instructions for Mounting Arm ... 88
 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
Set Zero soft key ... 28
Setup ... 23
Setup soft key ... 23
Soft key Labels ... 16
Specs. for Milling ... 85
Specs. for Turning ... 86
Status Bar ... 16
Status Bar (setting) ... 25
Stopwatch (setting) ... 25

T

Taper calculator ... 64
Tool call ... 35
Tool setting, turning ... 59
Tool soft key ... 31
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

☎ +49 (8669) 5061

E-Mail: info@heidenhain.de

www.heidenhain.de

