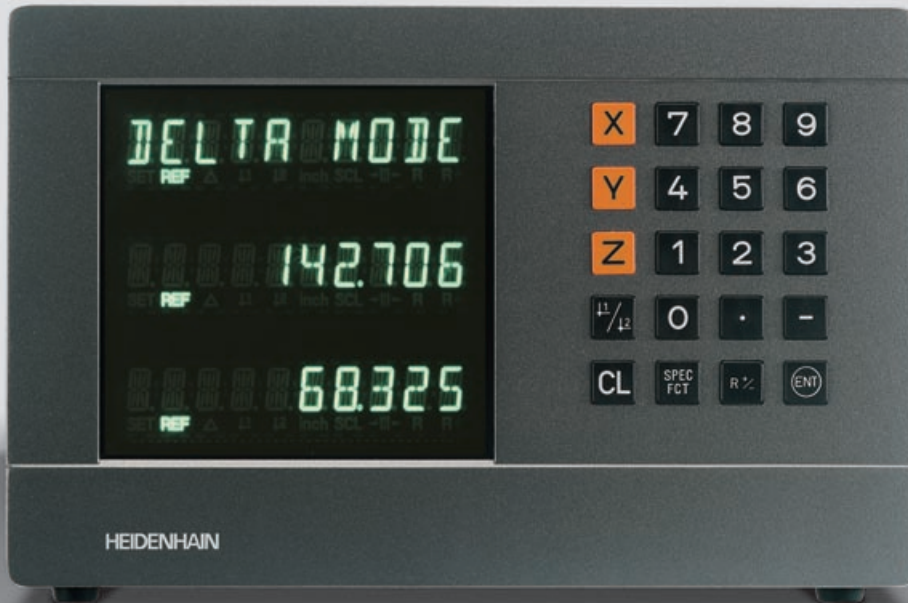




HEIDENHAIN



User's Manual

ND 710

ND 750

**Position Display Units
for Milling Machines**

9/2000

**Position display
(ND 710 only two axes)**

- Select coordinate axes
(ND 710 only X and Y)
- Select axis-specific operating parameters

Status display:

SET = Datum setting

REF = blinking:

Traverse the
reference points.
On continuously:
Reference points
have been traversed.

Δ = Distance-to-go display

1 2 Datum 1 or 2

Inch = Display in inches

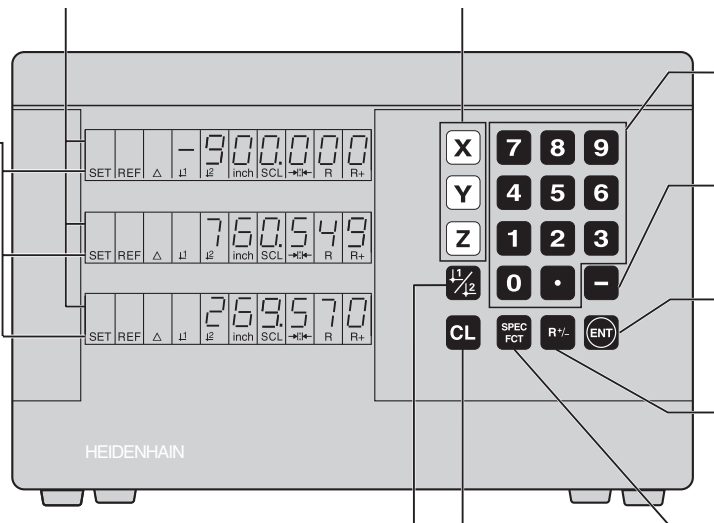
SCL = Scaling factor

->||<- = Touching the edge /
centerline

R = Radius/diameter
display

R+/- = Radius compensation

- Select datum 1 or 2
- Page backward in the list of
special functions
- Page backward in the list of
parameters



Numerical input

- Change the algebraic sign
- Call the last dialog
- In the parameter list:
change parameters

- Confirm entry
- In the parameter list
page forward

**Call radius compensation
of the current tool**

- Select special functions
- In the list of special functions
page forward

- Cancel entry
- Reset the operating mode
- Zero the selected axis
(if activated in P80)
- Select parameters
CL plus two-digit number



This manual is for the ND display units with the following software numbers or higher:

ND 710 for two axes
ND 750 for three axes

246 271-06
246 271-06

About this manual

This manual is divided into two parts:

Part I: Operating Instructions

- Fundamentals of positioning
- ND functions

Part II: Installation and Specifications

- Mounting the display unit on the machine
- Description of operating parameters

Part I Operating Instructions

Fundamentals	4
Switch-On, Traversing the Reference Points	9
Datum Setting	10
Tool Compensation	19
Moving the Axes with Distance-To-Go	20
Bolt Hole Circles and Bolt Hole Circle Segments	22
Linear Hole Patterns	25
Working with a Scaling Factor	28
Error Messages	29

Part II Installation and Specifications

Page 31

Fundamentals



You can skip this chapter if you are already familiar with coordinate systems, incremental and absolute dimensions, nominal positions, actual positions and distance-to-go.

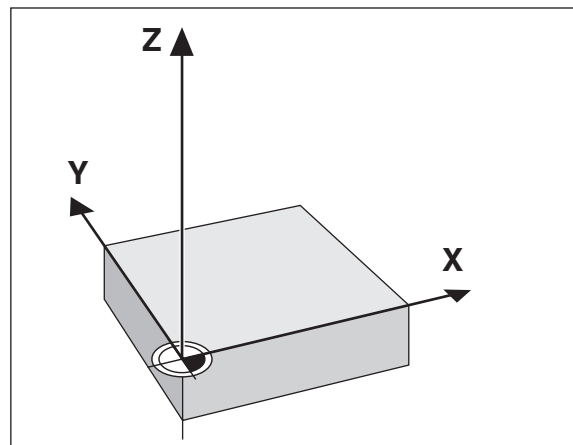
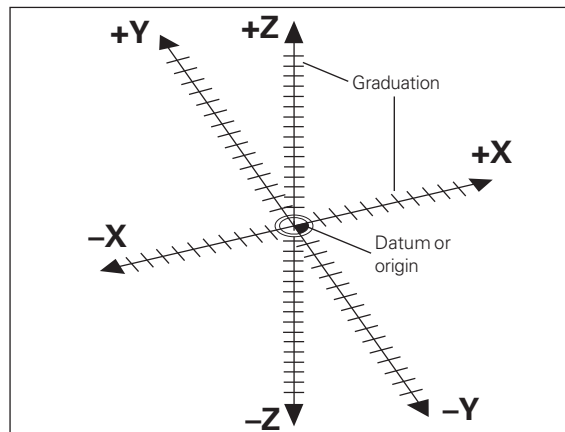
Coordinate system

To describe the geometry of a workpiece, the *Cartesian** coordinate system is used. The Cartesian coordinate system consists of three mutually perpendicular axes X, Y and Z. The point of intersection of these axes is called the **datum** or origin of the coordinate system.

Think of the axes as scales with divisions (usually in millimeters) which allow us to fix points in space referenced to the datum.

To determine positions on a workpiece, the coordinate system is "laid" onto the workpiece.

The machine axes are parallel to the axes of the coordinate system. The Z axis is normally the tool axis.



¹⁾ Named in honor of the French mathematician and philosopher René Descartes (1596 to 1650)

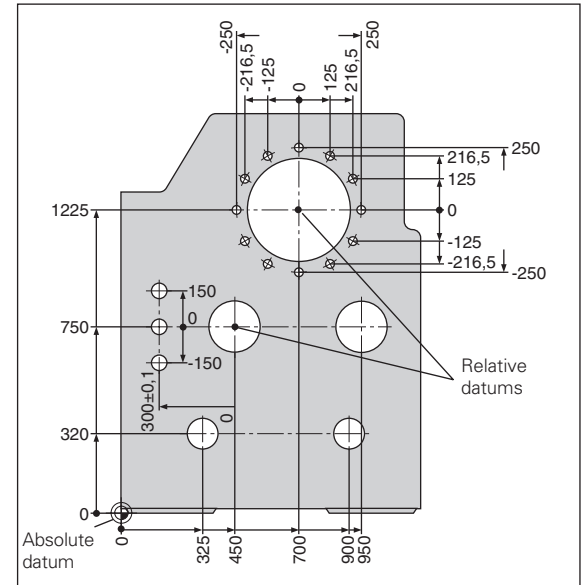
Datum setting

The workpiece drawing is used as the basis for machining the workpiece. To enable the dimensions in the drawing to be converted into traverse distances of machine axes X, Y and Z, each drawing dimension requires a datum or reference point on the workpiece (since a position can only be defined in relationship to another position).

The workpiece drawing always indicates **one** absolute datum (the datum for absolute dimensions). However, it may contain additional relative datums.

In the context of a numerical position display unit, *datum setting* means bringing the workpiece and the tool into a defined position in relation to each other and then setting the axis displays to the value which corresponds to that position. This establishes a fixed relationship between the actual positions of the axes and the displayed positions.

You can set 2 absolute datum points and store them in nonvolatile memory.



Absolute workpiece positions

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

Example Absolute coordinates of position ①:

$$\begin{aligned} X &= 10 \text{ mm} \\ Y &= 5 \text{ mm} \\ Z &= 0 \text{ mm} \end{aligned}$$

If you are working according to a workpiece drawing with absolute dimensions, then you are moving the tool **to** the coordinates.

Relative workpiece positions

A position can also be defined relative to the previous nominal position. The datum for the dimension is then located at the previous nominal position. Such coordinates are termed **relative coordinates** or chain dimensions. Incremental coordinates are indicated by a preceding **I**.

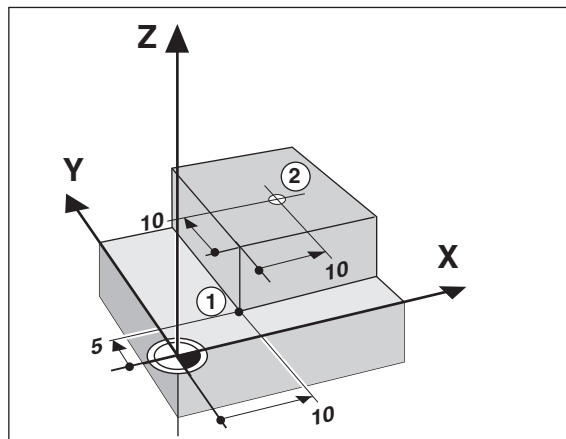
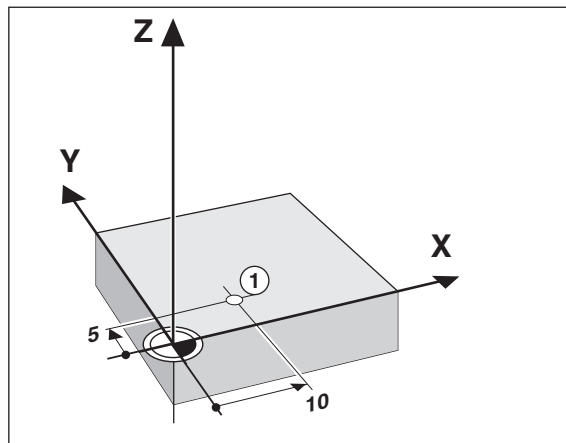
Example Relative coordinate of position ② referenced to position ①:

$$\begin{aligned} IX &= 10 \text{ mm} \\ IY &= 10 \text{ mm} \end{aligned}$$

If you are working according to a workpiece drawing with incremental dimensions, then you are moving the tool **by** the dimensions.

Sign for incremental dimensioning

A relative dimension has a **positive** sign when the axis is moved in the positive direction, and a **negative** sign when it is moved in the negative direction.



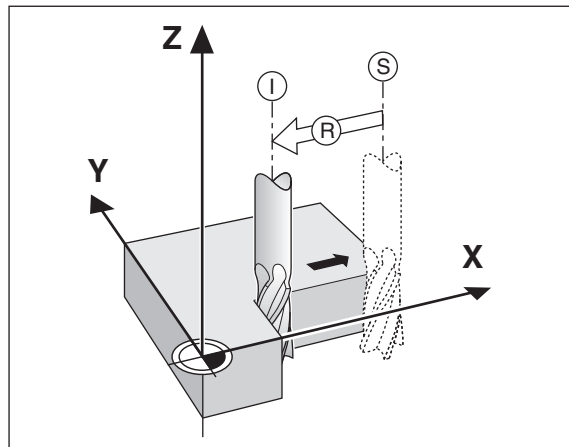
Nominal position, actual position and distance-to-go

The position to which the tool is to move is called the **nominal** position (Ⓢ). The position at which the tool is actually located at any given moment is called the **actual** position (Ⓜ).

The distance from the nominal position to the actual position is called the distance-to-go (Ⓡ).

Sign for distance-to-go

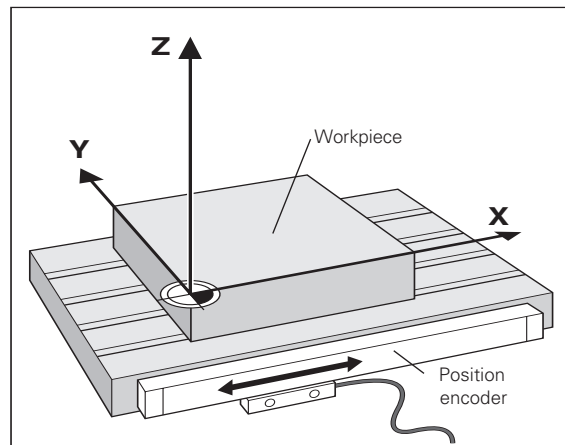
When you are using the distance-to-go display, the nominal position becomes the relative datum (display value 0). The distance-to-go is therefore negative when you move in the positive axis direction, and positive when you move in the negative axis direction.



Position encoders

The position encoders on the machine convert the movements of the machine axes into electrical signals. The ND display unit evaluates these signals, determines the actual position of the machine axes and displays the position as a numerical value.

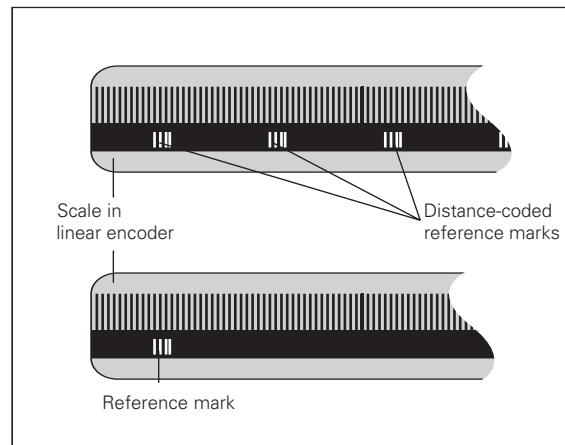
If the power is interrupted, the relationship between the machine axis positions and the calculated actual positions is lost. The reference marks on the position encoders and the REF reference mark evaluation feature enable the ND to quickly re-establish this relationship again when the power is restored.



Reference marks

The scales of the position encoders contain one or more reference marks. When a reference mark is crossed over, a signal is generated which identifies that position as a reference point (scale datum = machine datum).

When this reference mark is crossed over, the ND's reference mark evaluation feature (REF) restores the relationship between axis slide positions and display values which you last defined by setting the datum. If the linear encoders have **distance-coded** reference marks, you only need to move the machine axes a maximum of 20 mm to do this.



Switch-On, Traversing the Reference Marks

0 → 1

Turn on power (switch located on rear panel).
REF and decimal points in status display blink.

ENT ... CL



Confirm reference traverse mode. REF remains
on continuously. Decimal points blink.



Cross over the reference marks in all axes (in any
sequence). Each axis display becomes active
when its reference mark is crossed over.

Crossing over the reference marks stores the last relationship
between axis slide positions and display values for datum points 1 and
2 in nonvolatile memory.

Note that if you choose *not* to traverse the reference marks (by
clearing the dialog ENT ... CL with the CL key), this relationship will be
lost if the power is interrupted or when the unit is switched off.



If you wish to use multipoint axis error compensation you
must traverse the reference marks (see "Multipoint axis error
compensation")!

Datum Setting



If you want to save the datum points in nonvolatile memory, you must first cross over the reference marks.

Only after crossing over the reference marks can you set new datums or activate existing ones.

In P70, you can select:

- Two datum points: The selected datum is displayed via 1 or 2
- Nine datum points: The selected datum is displayed in the lowest axis via d1 to d9.

There are several ways to set datums:

Touch the edge of the workpiece with the tool and then set the desired datum. You can also touch two edges and set the centerline between them as a datum, or touch the inside of a circle and set the circle center as a datum (see examples). The tool data of the tool used for this are automatically considered (see "Tool Compensation").

To call a datum point you have set, proceed as follows:

You have set two datum points in P70:



Select datum 1 or 2.

You have set nine datum points in P70:



Press the datum key ("d" blinks).



Enter a datum number (1 to 9).

Probing a workpiece edge to find a datum

The ND display units support the following probing functions:

- “PROBE EDGE” Setting a workpiece edge as datum.
- “PROBE MIDPOINT” Setting a midpoint between two workpiece edges as datum.
- “PROBE CIRCLE” Setting the center of a circle as datum.

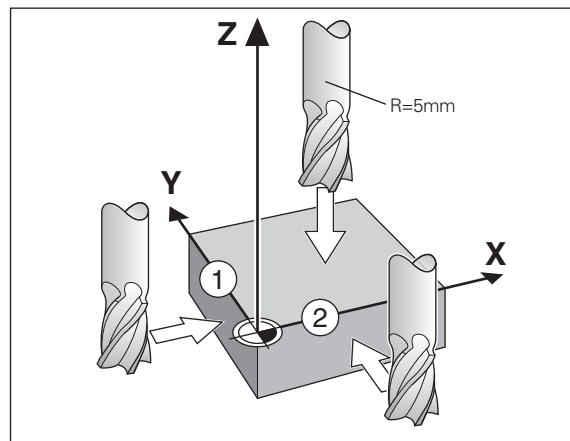
The probing functions are accessible in the SPEC FCT mode of operation.

The functions “PROBE EDGE”, “PROBE MIDPOINT” and “PROBE CIRCLE” are described on the following pages.

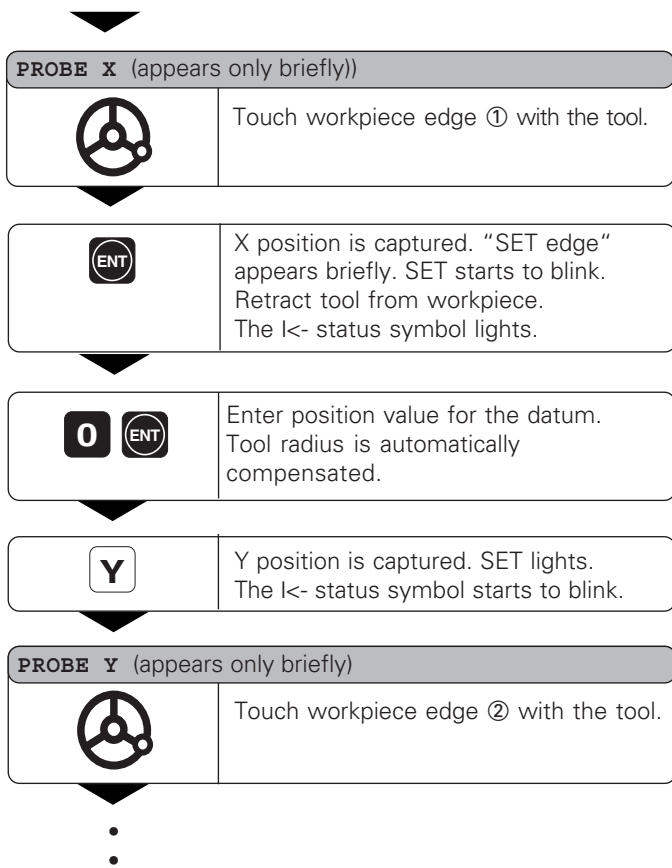
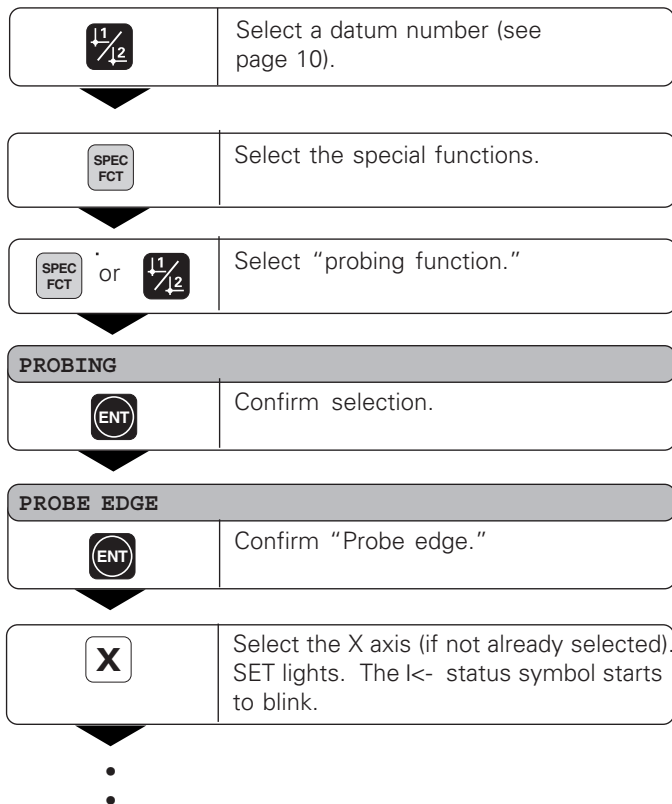
Datum setting with the tool

Example:


Working plane	X / Y
Tool axis	Z
Tool radius	R = 5 mm
Axis sequence for datum setting	X – Y – Z





Probing a workpiece edge to find a datum




↓

	Y position is captured. "SET edge" appears briefly. SET starts to blink. Retract tool from workpiece. The I<- status symbol lights.
---	---


↓

 	Enter position value for the datum in the Y axis. Tool radius is automatically compensated.
---	---


↓

	Select the Z axis. SET lights. The I<- status symbol starts to blink. ¹⁾
---	---

↓



PROBE Z (appears only briefly)	
	Touch workpiece surface with the tool.

↓



	Z position is captured. "SET edge" appears briefly. SET starts to blink. Retract tool from workpiece. The I<- status symbol lights.
---	---

⋮

↓

 	Enter position value for the datum in the Z axis.
---	---

↓






 or 	After setting the datum, exit the probing functions.
--	--

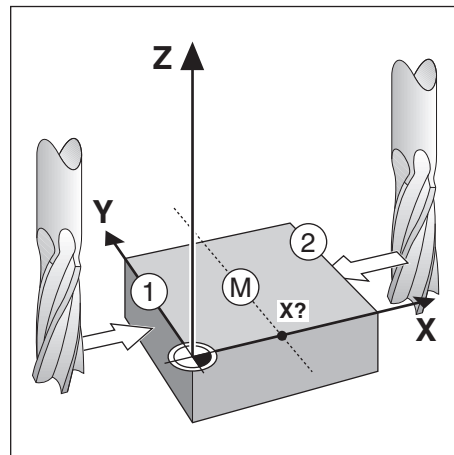
¹⁾ only with ND 750

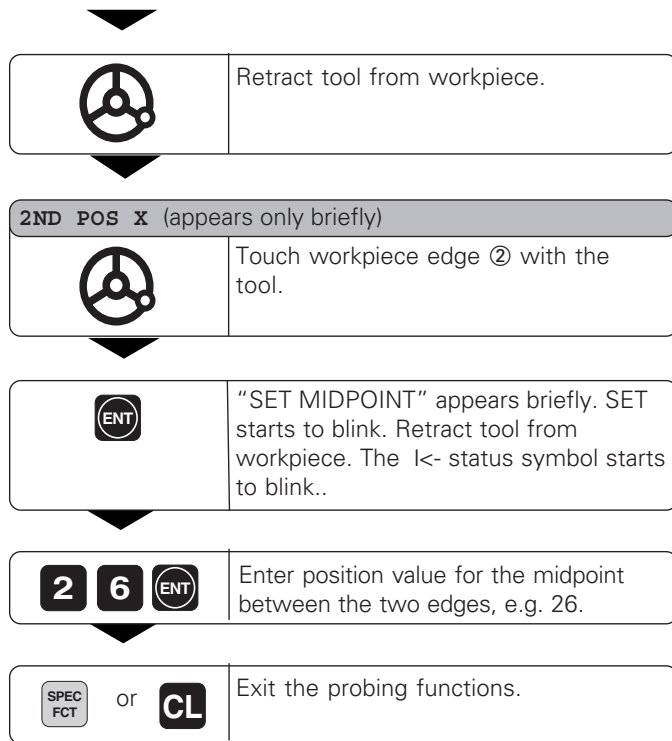
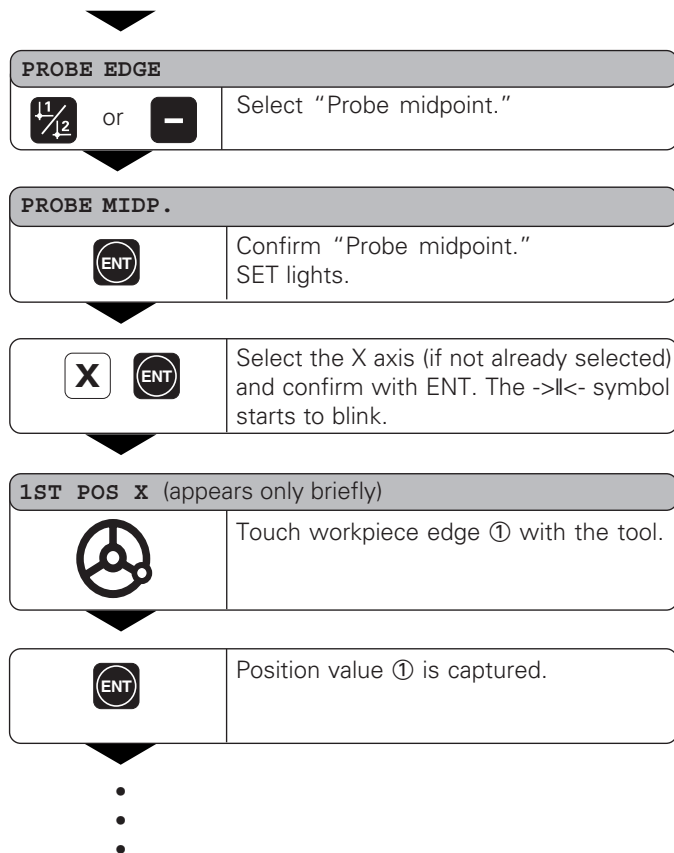
Probing workpiece edges to find a midpoint datum

The edges to be probed run parallel to the Y axis.

Follow the procedure below for all midpoints between two edges:


	Select a datum number (see page 10).
	Select the special functions.
 or 	Select the probing function.
PROBING	
	Confirm selection.
⋮	









Probing the inside of a circle to set its center as datum



To determine the circle center, the tool must probe four points. The points to be probed are located in the X-Y plane.

	Select a datum number (see page 10).
---	--------------------------------------

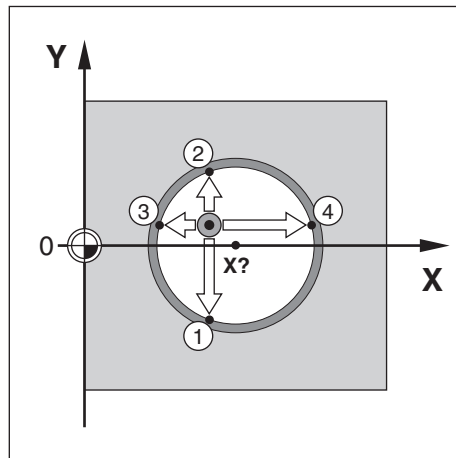
	Select the special functions.
---	-------------------------------

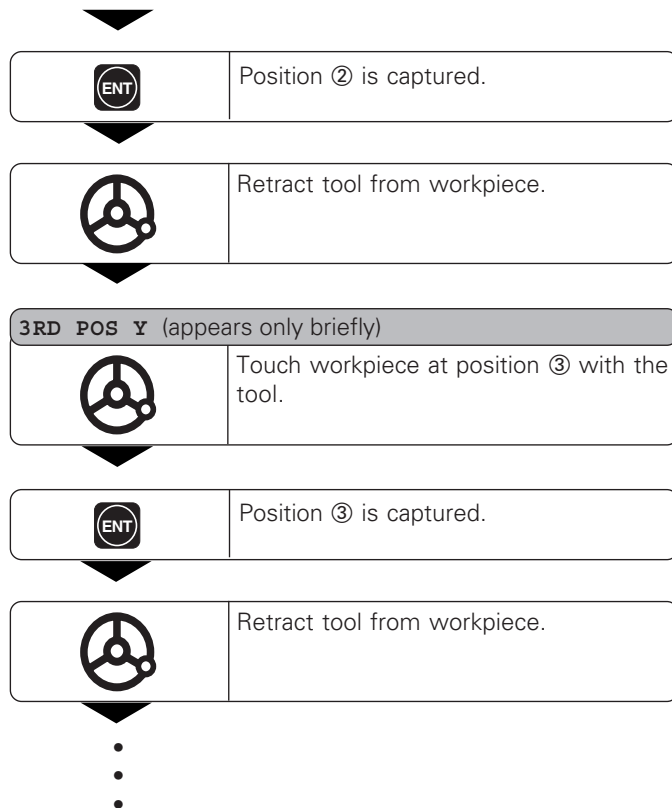
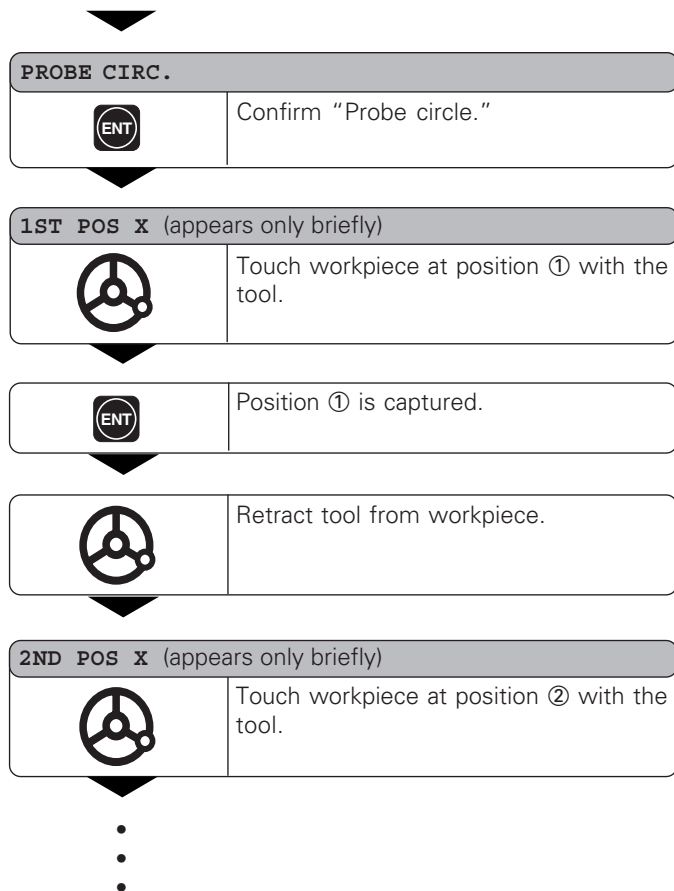
 or 	Select the probing function.
--	------------------------------

PROBING	
	Confirm selection.

PROBE CIRC.	
 or 	Select "Probe circle."

⋮





4TH POS Y (appears only briefly)



Touch workpiece at position ④ with the tool.



Position ④ is captured. "CENTER X" appears briefly. Status display SET starts to blink in the X axis.

2 6 ENT

Enter position value for the X coordinate of circle center, e.g. 26. "CENTER Y" appears briefly. Status display SET starts to blink in the Y axis.

0 ENT

Enter position value for the Y coordinate of circle center, e.g. 0.

SPEC
FCT

or

CL

Exit the probing functions.

Tool Compensation

You can enter the axis, length and diameter of the current tool.

SPEC FCT	Select the special functions.
-----------------	-------------------------------

SPEC FCT or $\frac{1}{2}$	Select "tool data."
--	---------------------

TOOL DATA	
ENT	Confirm tool data input mode.

TOOL DIAM.	
2 0 ENT	Enter the tool diameter, e.g. 20 mm, and confirm with ENT.

TOOL LENGTH	
5 0 ENT	Enter the tool length, e.g. 50 mm, and confirm with ENT.

⋮

¹⁾ only with ND 750

TOOL AXIS	
Z	Set the tool axis.

TOOL AXIS	
SPEC FCT or CL	Exit the special functions.

Moving the Axes with Distance-To-Go Display

Normally, the display shows the actual position of the tool. However, it is often more helpful to display the distance remaining to the nominal position (the distance-to-go). You can then position simply by moving the axis until the display value is zero.

You can enter the absolute coordinates in the distance-to-go display. An active radius compensation will be considered.

Example: Milling a shoulder with distance-to-go

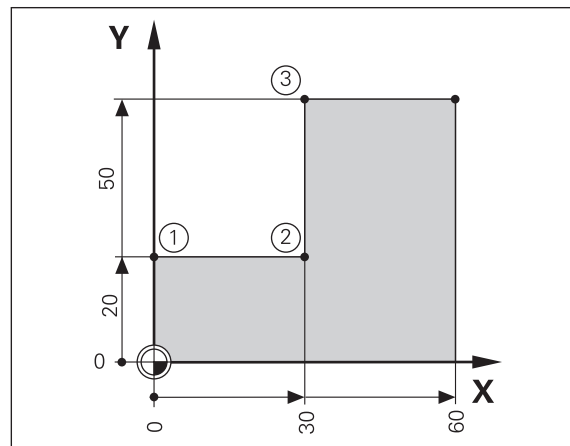
SPEC FCT	Select the special functions.
-----------------	-------------------------------

SPEC FCT or $\frac{1}{\sqrt{2}}$	Select "delta mode."
---	----------------------

DELTA MODE	
ENT	Confirm your selection, the Δ symbol lights.

Y 20 R+	Select the axis, enter the nominal value, e.g. 20 mm, select radius compensation R+, confirm with ENT.
ENT	

...





Move the machine axis to zero ①.



X

3

0

R⁺/₋

R⁺/₋

ENT

Select the axis, enter the nominal value, e.g. 30 mm, select radius compensation R⁻, and confirm with ENT.



Move the machine axis to zero ②.



Y

5

0

R⁺/₋

ENT

Select the axis, enter the nominal value, e.g. 50 mm, select radius compensation R⁺, confirm with ENT.



Move the machine axis to zero ③.



**SPEC
FCT**

ENT

or

CL

If appropriate, switch off the distance-to-go display.

Bolt Hole Circles and Bolt Hole Circle Segments

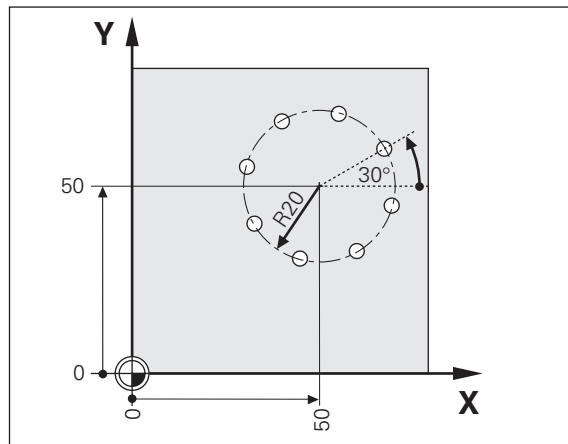
Your display unit enables you to quickly and easily drill bolt hole circles and bolt hole circle segments. The required data is requested in the message field.

Each hole can be moved to by traversing to display value zero. This requires entry of the following data:

- Number of holes (maximum: 999)
- Circle center
- Circle radius
- Starting angle for first hole
- Angle step between the holes (only for circle segments)
- Hole depth

Example

Number of holes	8
Coordinates of the center	X = 50 mm
	Y = 50 mm
Circle radius	20 mm
Starting angle	30 degrees
Hole depth	Z = -5 mm



SPEC FCT	Select the special functions.
-----------------	-------------------------------

SPEC FCT or 1/2	Select "bolt hole" circle.
-------------------------------	----------------------------

BOLT HOLE	
ENT	Confirm your selection.

FULL CIRCLE	
if req. - ENT	Confirm "full circle."

NUMB. HOLES	
8 ENT	Enter the number of holes, e.g. 8. Confirm with ENT.

⋮

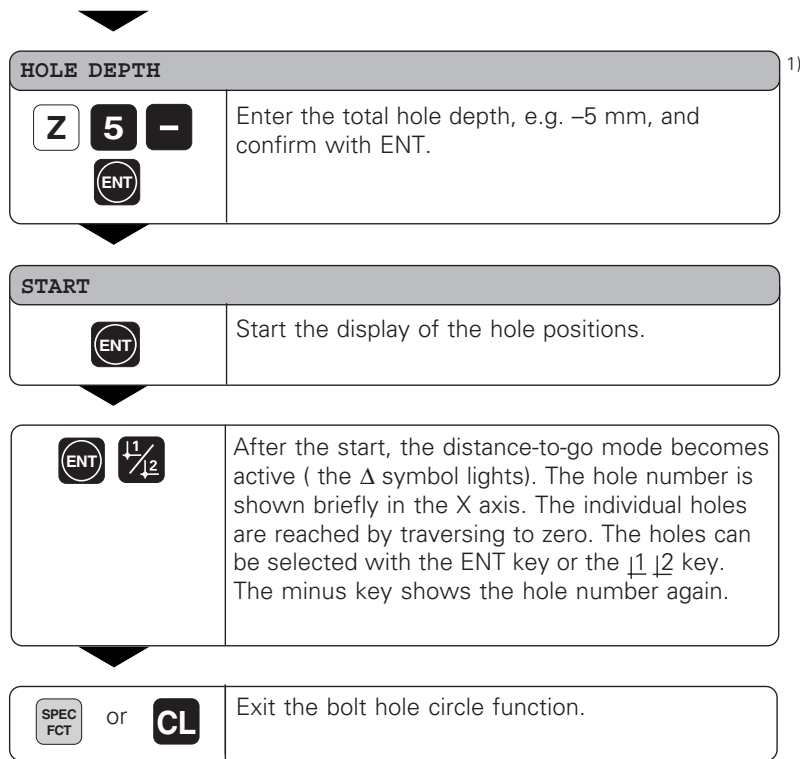
CENTER X	
X 5 0 ENT	Enter the X coordinate of circle center, e.g. 50 mm, confirm with ENT.

CENTER Y	
Y 5 0 ENT	Enter the Y coordinate of circle center, e.g. 50 mm, confirm with ENT.

RADIUS	
2 0 ENT	Enter the radius of the bolt hole circle, e.g. 20 mm. Confirm with ENT.

START ANGLE	
3 0 ENT	Enter the start angle for the first hole, e.g. 30°. Confirm with ENT.

⋮



1) only with ND 750

Linear Hole Patterns

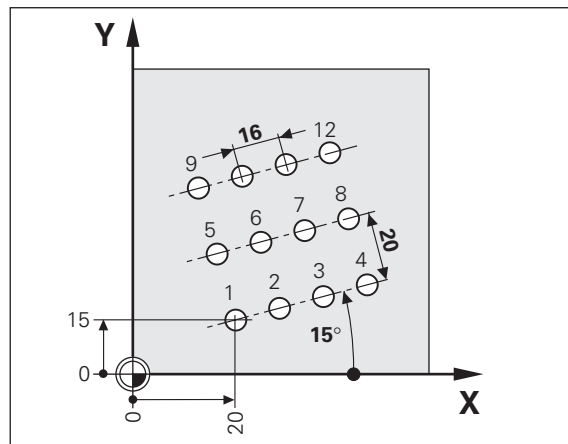
The linear hole pattern feature allows you to easily create rows of holes to cover an area. The required data are requested in the message field.

You can position to each hole by traversing to display value zero. The following data are required:

- Coordinates of the first hole
- Number of holes per row (maximum: 999)
- Spacing between holes
- Angle between the rows and the reference axis
- Hole depth
- Number of rows (maximum: 999)
- Spacing between rows

Example

Coordinates of the first hole	X = 20 mm
	Y = 15 mm
Number of holes per row	4
Spacing between holes	16 mm
Angle	15 degrees
Hole depth	Z = -30 mm
Number of rows	3
Spacing between rows	20 mm

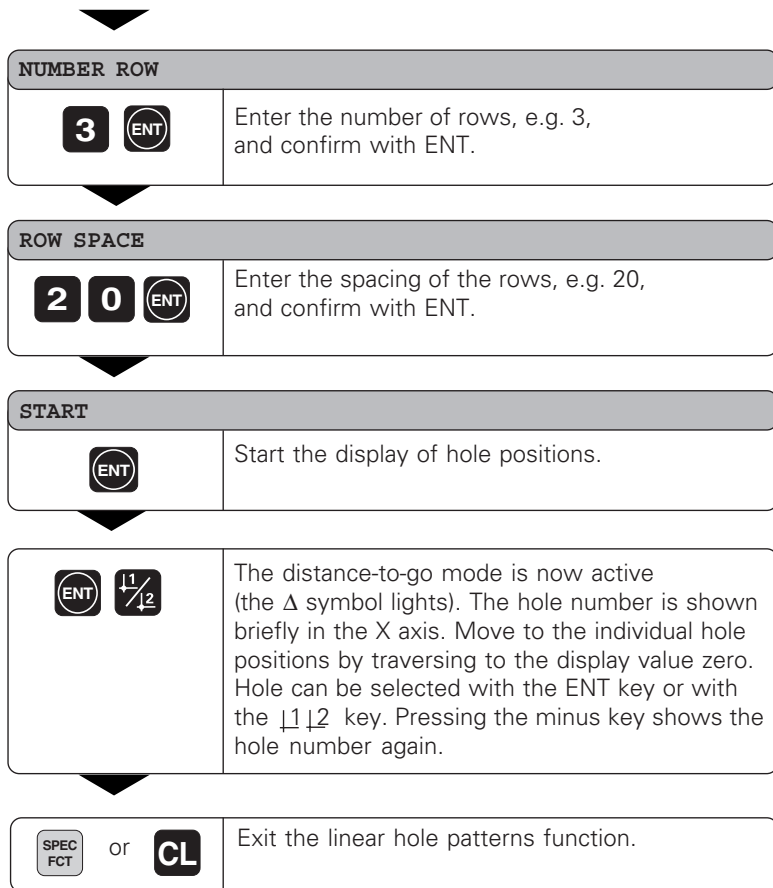


SPEC FCT	Select the special functions.
SPEC FCT or 1 1/2	Select "hole pattern."
LIN. HOLE	
ENT	Confirm "linear hole" pattern.
1ST HOLE X	
2 0 ENT	Enter the X coordinate of the first holes, e.g. 20, and confirm with ENT.
1ST HOLE Y	
1 5 ENT	Enter the Y coordinate of the first holes, e.g. 15, and confirm with ENT.
...	

HOLES ROW	
4 ENT	Enter the number of holes per row, e.g. 4, and confirm with ENT.
HOLE SPACE	
1 6 ENT	Enter the spacing between holes in the row, e.g. 16, and confirm with ENT.
ANGLE	
1 5 ENT	Enter the angle, e.g. 15 degrees, and confirm with ENT.
HOLE DEPTH	
3 0 - ENT	Enter the hole depth, e.g. -30 mm, and confirm with ENT.
...	

1)

1) only with ND 750



Working with a Scaling Factor

Scaling factors enable you to increase or decrease the display values based on the actual traverse distance. The display values are changed symmetrically about the datum.

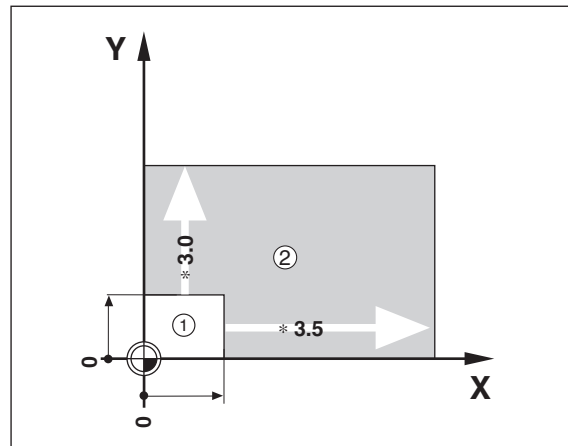
Enter scaling factors separately for each axis in parameter P12.

Parameter P11 activates and deactivates the scaling factors in all axes (see "Operating Parameters").

Example for enlarging a workpiece:

P12.1	3.5
P12.2	3.0
P11	ON

This results in a larger workpiece as shown in the illustration at right:
① is the original size, ② is with axis-specific scaling factors.



If a scaling factor is active, SCL lights in the status display.

Error Messages

Message	Cause and effect
SIGNAL X	Encoder signal is too small, e.g. when an encoder is contaminated.
PROB. ERROR	Before touching off on the workpiece, the tool must move by a distance of at least 0.2 mm.
ERR. REF. X	The spacing of the reference marks as defined in P43 is not the same as the actual spacing.
FRQ. ERR. X	The input frequency for this encoder input is too high. This can occur when the scale is moved too fast.
ERR. MEMORY	Check sum error: Check the datum, operating parameters and compensation values for multi-point axis error compensation. If the error recurs, contact your service agency!

To erase error messages:

After you have removed the cause of error:

- Press the CL key.

Part II Installation and Specifications

Items Supplied	32
Connections on Rear Panel	33
Mounting	34
Power Connection	34
Connecting the Encoders	35
Operating Parameters	36
Entering/changing operating parameters	36
Operating parameter list	37
Linear Encoders	39
Setting the display step	39
Display step, signal period, and subdivision	39
Parameter settings for HEIDENHAIN linear encoders with 11 μ App	40
Multipoint Axis Error Compensation	41
Specifications	44
Dimensions of the ND 710/ND 750	45

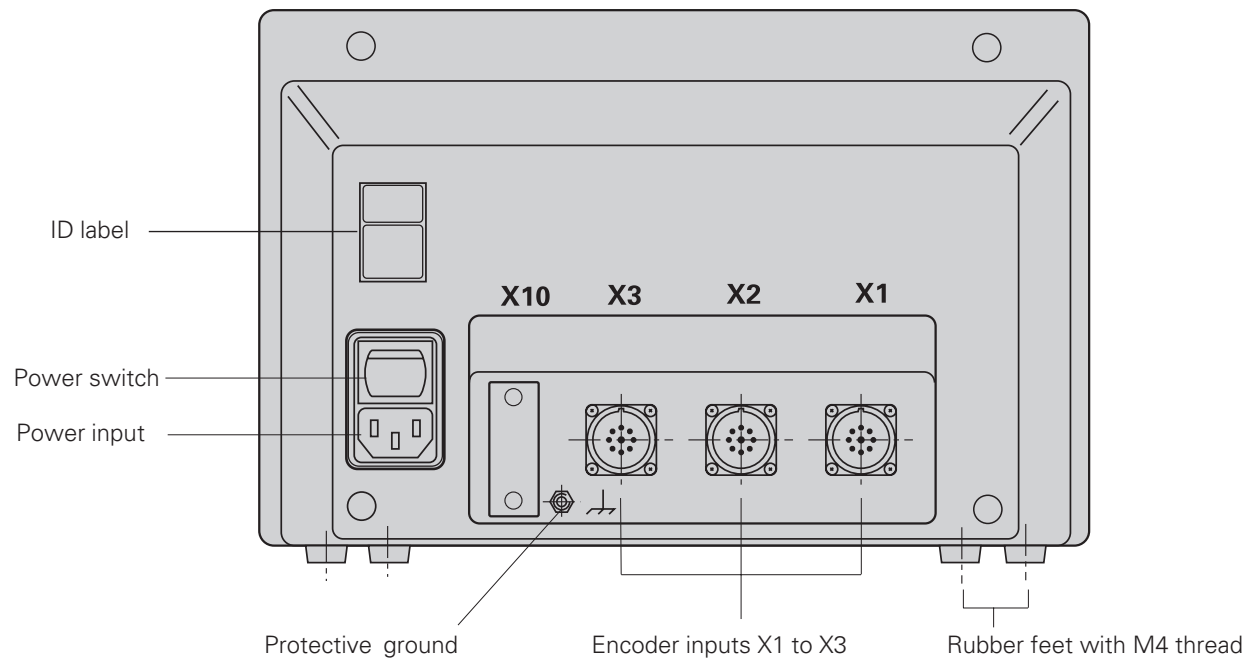
Items Supplied

- **ND 710** for 2 axes
or
- **ND 750** for 3 axes
- **Power connector** Id. Nr. 257 811-01
- **User's Manual**

Optional Accessories

- **Tilting base** for housing bottom
Id. Nr. 281 619-01

Connections on Rear Panel

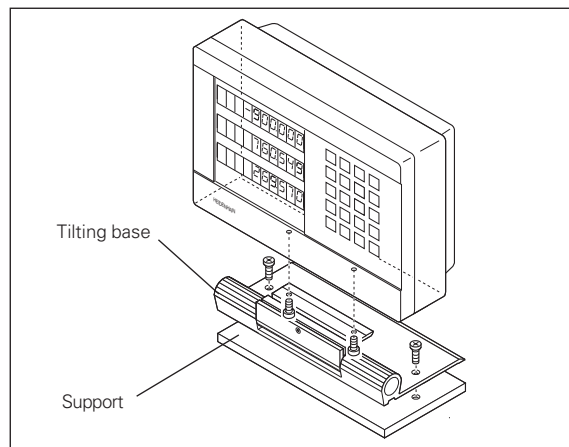


The interfaces X1, X2, X3 comply with the requirements for electrical separation according to EN 50178!

Mounting

ND 710/ND 750

To mount the display unit on a support, use the M4 threaded holes in the rubber feet. You can also mount the display unit on the optional tilting base.



Power Connection

Power leads: (L) and (N)

Connect protective ground to (⊥)!

Power supply: 100 Vac to 240 Vac (-15 % to +10 %)
50 Hz to 60 Hz (± 2 Hz)

A voltage selector is not necessary.



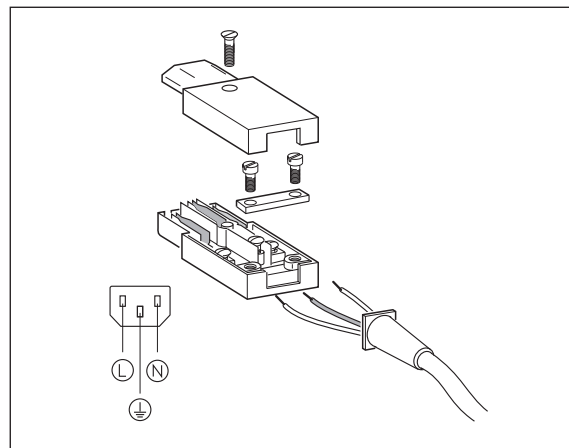
• Danger of electrical shock!

Connect a protective ground. This connection must never be interrupted.

- Unplug the power cord before opening the housing.



To increase the noise immunity, connect the ground terminal on the rear panel to the central ground point of the machine. (Minimum cross-section: 6 mm²).



Connecting the Encoders

Your display unit will accept all HEIDENHAIN linear encoders with sinusoidal output signals (7 to 16 μA_{pp}) and distance-coded or single reference marks.

Assignment of the encoder inputs

Encoder input X1 is for the X axis.

Encoder input X2 is for the Y axis.

Encoder input X3 is for the Z axis (ND 750 only).

Encoder monitoring system

Your display unit features a monitoring system for checking the amplitude and frequency of the encoder signals. If it detects a faulty signal, one of the following error messages will be generated:

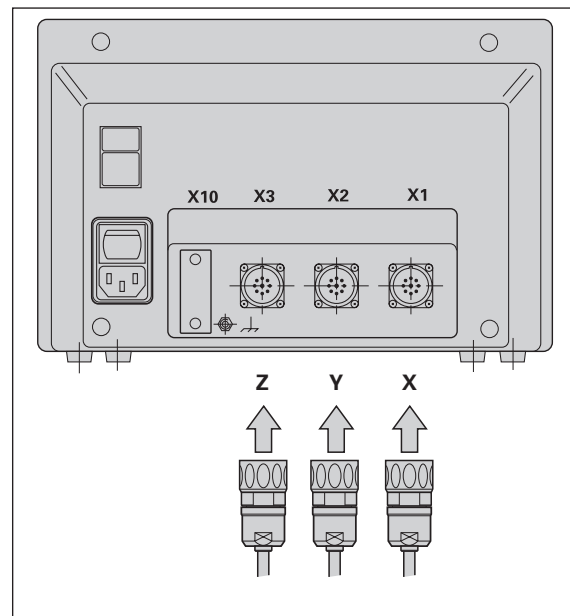
SIGNAL X

FRQ. X

Encoder monitoring can be activated with parameter P45.

If you are using linear encoders with distance-coded reference marks, the encoder monitoring system also checks whether the spacing of the reference marks as defined in parameter P43 is the same as the actual spacing on the scales. If it is not, the following error message will be generated:

ERR. REF. X



Operating Parameters

Operating parameters allow you to modify the operating characteristics of your display unit and define the evaluation of the encoder signals. Operating parameters that can be changed by the user are called user parameters, and can be accessed with the SPEC FCT key and the dialog "PARAMETER" (user parameters are identified as such in the parameter list). The full range of parameters can only be accessed through the dialog "CODE" and by entering 95148. Operating parameters are designated by the letter P and a number. Example: **P11**. The parameter designation is shown in the input field when you select it with the DATUM and ENT key in the X display. The parameter setting is shown in the Y display.

Some operating parameters have separate values for each axis. In the **ND 750**, these parameters are identified by an index of 1 to 3, and in the **ND 710** by an index of one to two.

Example: P12.1 scaling factor, X axis
 P12.2 scaling factor, Y axis
 P12.3 scaling factor, Z axis (ND 750 only)

The operating parameters are preset before the unit leaves the factory. These factory settings are indicated in the parameter list in **boldface type**.

Entering and changing operating parameters

To access the operating parameters

- Press the SPEC FCT key.
- Press the SPEC FCT key or 1 2 , until "PARAMETER" appears in the X display.
- Confirm your selection by pressing "ENT."

To select protected operating parameters

- Press the 1 2 key to select user parameter P00 CODE.
- Enter the code number 95148.
- Confirm with "ENT."

To page through the operating parameters

- Page forwards by pressing the ENT key.
- Page backwards by pressing the 1 2 key.

To change parameter settings

- Press the minus key or enter the value and confirm with the ENT key.

To correct an entry

- Press CL: the old value reappears in the input line and becomes effective again.

To leave the operating parameters

- Press the SPEC FCT or CL key.

List of operating parameters

P00 CODE Enter the code number:

9 51 48:	Change protected operating parameters
66 55 44:	Display the software version (X display) Display date of release (Y display)
10 52 96:	Multipoint axis error compensation

P1 Unit of measure 1)

Display in millimeters	MM
Display in inches	INCH

P3.1 to P3.3 Radius/diameter display ¹⁾

Display position value as radius	RADIUS
Display position value as diameter	DIAMETER

P11 Activate scaling factor ¹⁾

Active	SCALING ON
Not active	SCALING OFF

P12.1 to P12.3 Define scaling factor ¹⁾

Enter a scaling factor separately for each axis:
 Entry value > 1: workpiece will "grow"
 Entry value = 1: workpiece will remain the same size
 Entry value < 1: workpiece will "shrink"
 Input range: 0.100000 to 9.999999
 Factory default setting: **1.000000**

P30.1 to P30.3 Counting direction

Positive counting direction with
 positive direction of traverse **DIRECT. POS**

Negative counting direction with
 positive direction of traverse **DIRECT. NEG**

P31.1 to P31.3 Signal period of the encoder

Input range: 0.00000001 to 99999.9999 µm
 Default setting: **20 µm**

P33.1 to P33.3 Counting mode

0 - 1 - 2 - 3 - 4 - 5 - 6 - 7 - 8 - 9
 0 - 2 - 4 - 6 - 8
 0 - 5

P38.1 to P38.3 Decimal places

1 / 2 / **3** / 4 (up to 6 with inch display)

P40.1 to P40.3 Select type of axis error compensation

No axis error compensation **CORR. OFF**

Linear error compensation active,
 multipoint error comp. not active **CORR. LIN**

Multipoint error compensation active,
 linear error compensation not active **CORR. ABS**

¹⁾ User parameter

P41.1 to P41.3 Linear axis error compensation

Input range (μm): -99999 to +99999
 Factory default setting: **0**

Example: Displayed length $L_a = 620.000$ mm
 Actual length (as determined for example with
 the VM 101 from HEIDENHAIN)
 $L_t = 619.876$ mm
 Difference $DL = L_t - L_a = -124$ μm
 Compensation factor k:
 $k = \Delta L / L_a = -124 \text{ μm} / 0.62 \text{ m} = -200$ [μm/m]

P43.1 to P43.3 Reference marks

One reference mark	SINGLE REF.M.
Distance-coded with 500 • SP	500 SP
Distance-coded with 1000 • SP	1000 SP
Distance-coded with 2000 • SP	2000 SP
Distance-coded with 5000 • SP	5000 SP

(SP: signal period)

P44.1 to P44.3 Reference mark evaluation

Evaluation	REF. X ON
No evaluation	REF. X OFF

P45.1 to P45.3 Encoder monitoring

Amplitude and frequency monitoring	ALARM ON
No monitoring	ALARM OFF

P48.1 to P48.3 Activate axis display

Axis display active	AXIS ON
Not active	AXIS OFF

P70 Number of datums

2 datums	2 DATUMS
9 datums	9 DATUMS

Function of the CL key

Reset to zero with CL	CL...RESET
No reset to zero with CL	CL.....OFF

P81 Function of the R+/- key

The R+/- key divides the actual value by two	R+/- 1/2
---	----------

No division of the actual value	R+/- OFF
---------------------------------	-----------------

P98 Dialog language¹⁾

German	LANGUAGE D
English	LANGUAGE GB
French	LANGUAGE F
Italian	LANGUAGE I
Dutch	LANGUAGE NL
Spanish	LANGUAGE E
Danish	LANGUAGE DK
Swedish	LANGUAGE S
Finnish	LANGUAGE FI
Czech	LANGUAGE CZ
Polish	LANGUAGE PL
Hungarian	LANGUAGE H
Portuguese	LANGUAGE P

¹⁾ User parameter

Linear Encoders

Selecting the display step with linear encoders

To select a certain display step, you must define the following operating parameters:

- Signal period (P31)
- Counting mode (P33)
- Decimal places (P38)

Example

Linear encoder with a signal period of 10 μm

Desired display step 0.000 5 mm

Signal period (P31) 10

Counting mode (P33) 5

Decimal places (P38) 4

The following tables will help you select the parameters.

Parameter settings for HEIDENHAIN linear encoders with 11 μA_{pp} signals

Model	Signal period [μm]	Reference marks	Millimeters			Inches		
			Display step [mm]	Count	Decimal places	Display step [inch]	Count	Decimal places
	P 31	P 43		P 33	P 38		P 33	P 38
CT	2	single	0.0005	5	4	0.00002	2	5
MT xx01			0.0002	2	4	0.00001	1	5
LIP 401A/401R		-/single	0.0001	1	4	0.000005	5	6
LF 103/103C	4	single/5000	0.001	1	3	0.00005	5	5
LF 401/401C			0.0005	5	4	0.00002	2	5
LIF 101/101C			0.0002	2	4	0.00001	1	5
LIP 501/501C								
MT xx	10	single	0.0005	5	4	0.00002	2	5
LS 303/303C	20	single/1000	0.01	1	2	0.0005	5	4
LS 603/603C			0.005	5	3	0.0002	2	4
LS 106/106C	20	single/1000	0.001	1	3	0.00005	5	5
LS 406/406C								
LS 706/706C								
ST 1201		-						
LB 302/302C	40	single/2000	0.005	5	3	0.0002	2	4
LIDA 10x/10xC			0.002	2	3	0.0001	1	4
LB 301/301C	100	single/1000	0.005	5	3	0.0002	2	4

Example:

Your encoder: LS 303 C, desired display step: 0.005 mm (5 μm), parameter settings:

P01 = mm, P43 = 1 000, P32 = 4, P33 = 5, P38 = 3

Multipoint Axis Error Compensation



If you want to use the multipoint axis error compensation feature, you must

- activate this feature with operating parameter 40 (see "Operating Parameters"),
- traverse the reference marks after switching on the display unit,
- enter compensation value table.

Your machine may have a non-linear axis error due to factors such as axis sag or drivescrew errors. Such deviations are usually measured with a comparator measuring system (such as the HEIDENHAIN VM 101).

For example, you can determine the screw pitch error $X=F(X)$ for the X axis.

An axis can only be corrected in relation to **one** axis that has an error. In each axis, a compensation value table with 64 compensation values can be generated. You can select the compensation table with the SPEC FCT key and the "PARAMETER\CODE" dialog.

To determine the compensation value (e.g. with a VM 101), the REF display must be selected after selecting the compensation-value table.



Select the REF.

The decimal point in the left display field indicates that the values displayed are referenced to the reference point. If the decimal point blinks, the reference marks have not been traversed.

Entries in the compensation value table

- Axis to be corrected: X, Y or Z (Z axis only with ND 750)
- Axis causing the error: X, Y or Z (Z axis only with ND 750)
- Datum for the axis to be corrected:
Here you enter the point starting at which the axis with error is to be corrected. This point indicates the absolute distance to the reference point.



Do not change the datum point after measuring the axis error and before entering the axis error into the compensation table.

- Spacing of the compensation points
The spacing of the compensation points is expressed as 2^x [μm].
Enter the value of the exponent x into the compensation value table.
Minimum input value: 6 (= 0.064 mm)
Maximum input value: 20 (= 8388.608 mm)
Example: 900 mm traverse and 15 compensation points:
results in 60.000 mm spacing between points.
Nearest power of two: 2^{16} [μm] = 65.536 mm
Entry in compensation value table: 16
- Compensation value
You enter the measured compensation value (in millimeters) for the displayed compensation point. Compensation point 0 always has the value 0 and cannot be changed.

Selecting the compensation table, entering an axis correction

SPEC FCT	Select special functions.
-----------------	---------------------------

SPEC FCT or 1/2	Select "parameter" if required, by repeatedly pressing the 1/2 key.
-------------------------------	--

PARAMETER	
ENT 1/2	Select dialog for entering the code number.

CODE	
1 0 5 2 9 6 ENT	Enter code number 105296 and confirm with ENT.

AXIS X	
X ENT	Select the axis to be corrected (e.g. X), and confirm with ENT.

X FCT. X	
X ENT	Enter the axis causing the error (e.g. X) (screw pitch error), and confirm with ENT.

⋮

DATUM X	
2 7 ENT	Enter the active datum for the error on the axis to be corrected (e.g. 27 mm) and confirm with ENT.

SPACING X	
1 0 ENT	Enter the spacing of the compensation points on the axis to be corrected, for example $2^{10} \mu\text{m}$ (equals 1.024 mm) and confirm with ENT.

27.000	
ENT 0 . 0 1 ENT	Compensation point no. 1 is displayed. Enter the associated compensation value (e.g. 0.01 mm) and confirm with ENT.

⋮

28.024



Enter all further compensation points. If you press the minus key, the unit will show the number of the current compensation point in the X display. Direct selection of compensation points: Press the minus key together with the number (two-digit) of the desired compensation point.

SPEC
FCT

or

CL

Conclude entry.

Deleting a compensation value table

SPEC
FCT

Select special functions.

SPEC
FCT

or

+/-

Select "parameter."

PARAMETER

ENT

+/-

Select the dialog for entering the code number.

CODE

1 0 5 2
9 6 ENT

Enter the code number 105296 and confirm with ENT.

AXIS X

Z

CL

Select the compensation value table (e.g., for the Z axis), and delete the table.

DELETE Z

ENT

Confirm with ENT, or cancel with CL.

SPEC
FCT

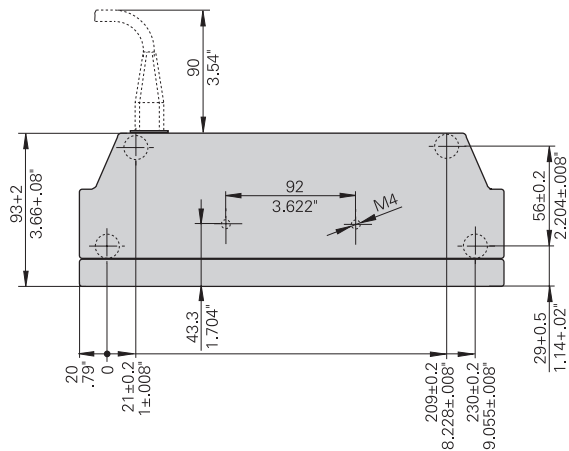
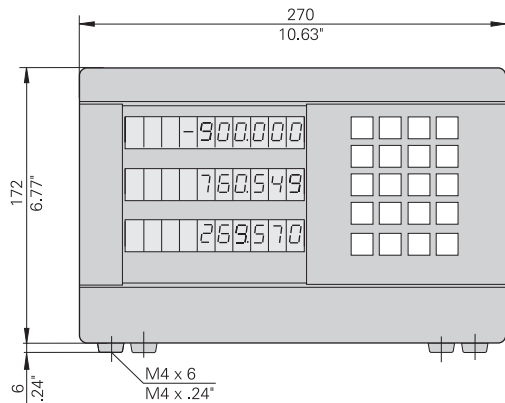
Conclude entry.

Specifications

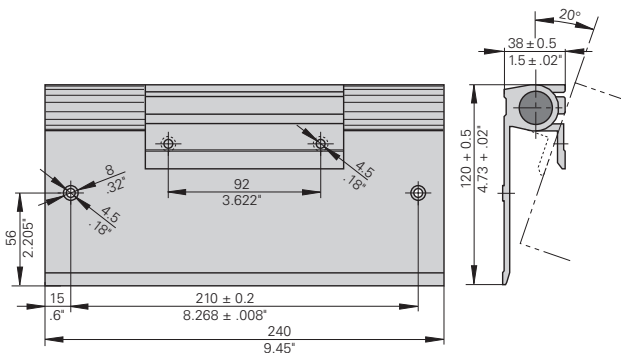
Housing	ND 710/ND 750 Bench-top design, cast-metal housing Dimensions (W • H • D) 270 mm • 172 mm • 93 mm
Oper. temperature	0° to 45° C (32° to 113° F)
Storage temperature	–20° to 70° C (–4° to 158° F)
Weight	Approx. 2.3 kg (5 lb)
Relative humidity	<75% annual average <90% in rare cases
Power supply	100 Vac to 240 Vac (–15% to +10%) 50 Hz to 60 Hz (± 2 Hz)
Power consumption	15 W
Protection	IP 40 as per IEC 529

Encoder inputs	For encoders with 7 to 16 μ App Grating period 2, 4, 10, 20, 40, 100, and 200 μ m Reference mark evaluation for distance-coded and single reference marks
Input frequency	Max. 100 kHz for 30 m cable length
Display step	Adjustable (see “Linear Encoders”)
Datums	2 (nonvolatile)
Functions	<ul style="list-style-type: none"> – Tool radius compensation – Distance-to-go display – Touching off function with tool – Circular & linear hole patterns – Scaling factor

Dimensions mm/inches



Tilting base



HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH

Dr.-Johannes-Heidenhain-Straße 5

83301 Traunreut, Germany

☎ + 49/86 69/31-0

FAX + 49/86 69/50 61

e-mail: info@heidenhain.de

☎ **Service** + 49/86 69/31-12 72

☎ TNC-Service + 49/86 69/31-14 46

FAX + 49/86 69/98 99

e-mail: service@heidenhain.de

<http://www.heidenhain.de>

HEIDENHAIN (G.B.) Limited

200 London Road, Burgess Hill

West Sussex RH15 9RD, Great Britain

☎ (0 14 44) 24 77 11

FAX (0 14 44) 87 00 24