HEIDENHAIN

User's Manual

POSITIP 880

POSITIP 880 Back View

Axis ports

Edge finder

Parallel port

Ground

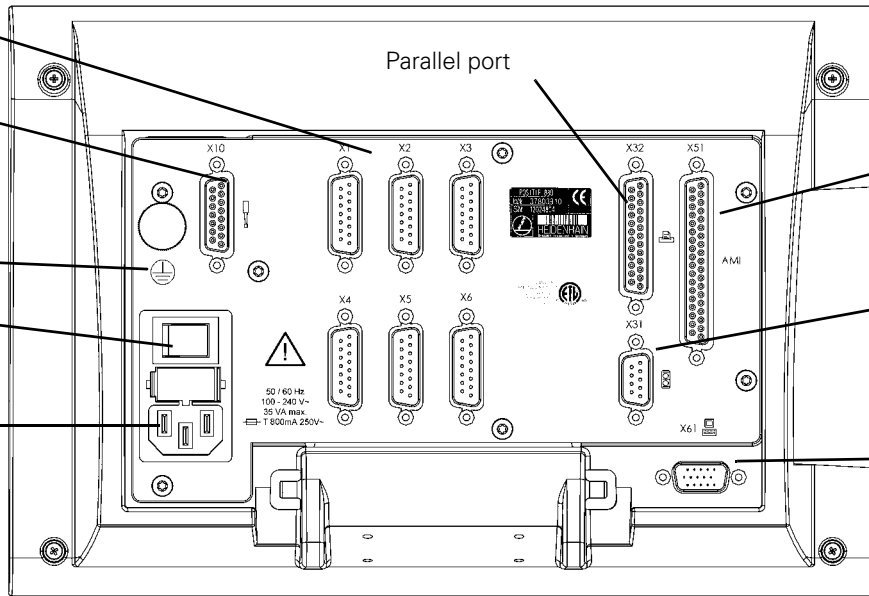Power button

Main power input

Auxiliary
Machine
Interface
connector

Serial
port

Remote
console

X10   X2   X3   X32   X51

X4   X5   X6   X31

AMI

P2ITIP 880

50 / 60 Hz
100 - 240 V~
35 VA max
T 800mA 250V~

X61

Keypad and soft keys

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

7	8	9
4	5	6
1	2	3
0	.	-
CE	I	(ENT)

Numeric input keys

Clear entry or
error messages

Change negative/positive
value key

Confirm entry

Incremental dimensions

Keys for selecting operating mode
(for detailed description of these key
functions, see Operating Modes in
chapter I-2)

Select entry fields

Soft keys - Row of keys under the screen
of the POSITIP whose functions vary
according to associated fields that appear
above them on the screen
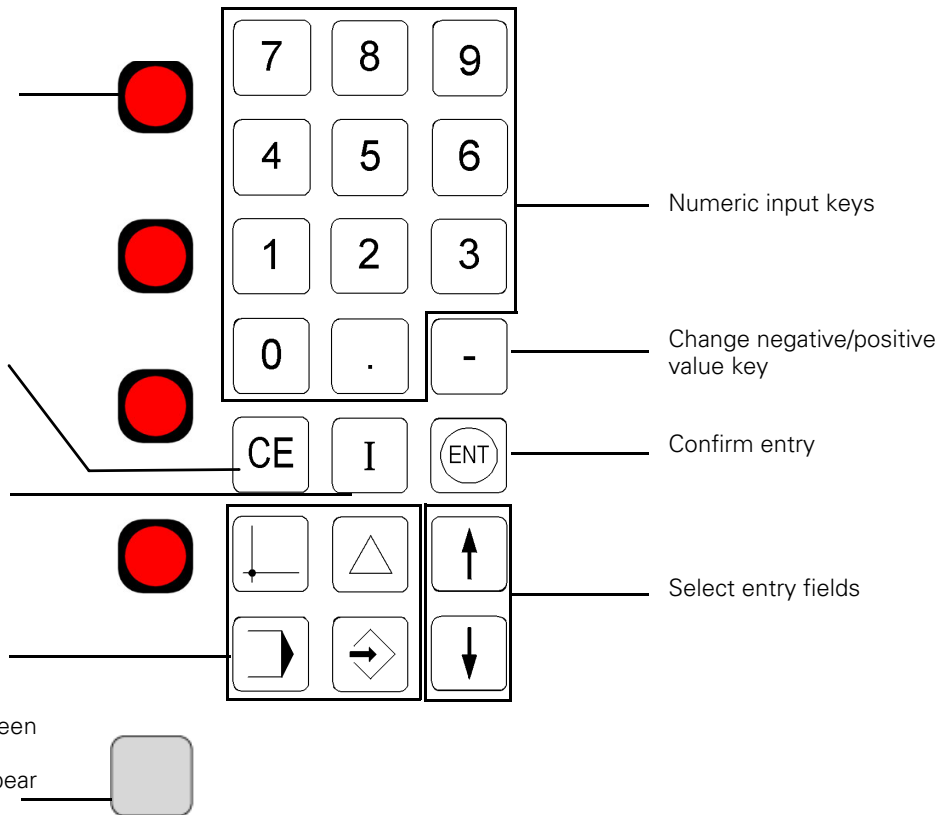
General Notes

Software version

The software version of your unit is shown on the initial power up screen and on the prompt bar after pressing the Help soft key.

☝ This User's Manual covers the functions of the POSITIP 880 for both **milling** and **turning** applications. Basic POSITIP 880 functions are covered in the first 4 chapters of this manual. The turning section deals only with the functions specific to turning applications.

About this manual

This manual is divided into two parts:

■ Part I: Operating Instructions
■ Part II: Technical Information

Operating Instructions

When using the POSITIP 880 in your work, you need only refer to the Operating Instructions (**Part I**).

I**f you're new to POSITIP 880**, you can use the operating instructions as a step-by-step workbook. This part begins with a short introduction to the basics of coordinate systems and position feedback, and provides an overview of the available features. Each feature is explained in detail, using an example which you can immediately try out on the machine — so you won't get "lost" in the theory. As a beginner you should work through all the examples presented.

If you're already familiar with POSITIP 880, you can use the Operating Instructions as a comprehensive review and reference guide.

Technical Information

If you are interfacing the POSITIP 880 to a machine or wish to use the data interfaces, refer to the technical information in **Part II**.

Dialog flowcharts

Dialog flowcharts are used for each example in this manual. They are laid out as follows:

PROMPT

KEY   This area explains the **key function** or **work step**. If necessary, supplementary information will also be included.

If there is an arrow at the end of the flowchart, this means that it continues on the next page.

A perforated line indicates an alternative method of carrying out the given function.

A **prompt** appears with some actions (not always) in the message bar on the screen.

Important Notes in this Manual

Special green note boxes contain especially important information. Please pay special attention to these notes. Neglecting this information can result in e.g. functions not working in the desired way or in causing damage to the workpiece or to the tool.

i

Symbols within the notes

Every note is marked with a symbol on the left informing about the meaning of the note.

General Information

e.g. on the behavior of the POSITIP 880.

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.

II Technical Information 111

Index

Operating Instructions

I – 1 Fundamentals of Positioning

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

Coordinate Systems

In order to define positions on a surface, a reference system is required.

For example, positions on the earth's surface can be defined absolutely by their geographic coordinates of longitude and latitude. In contrast to the relative definition of a position that is referenced to a known location, the network of horizontal and vertical lines on the globe constitute an absolute reference system. See Fig. I.1.
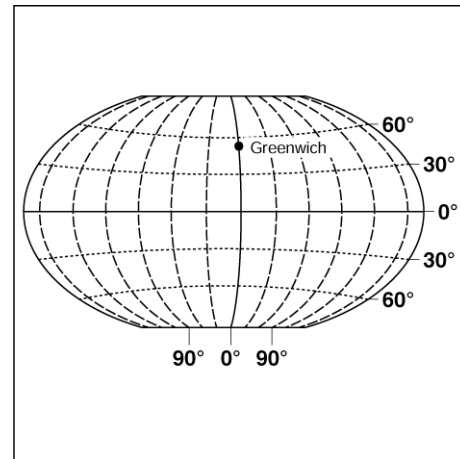
Fig. I.1   The geographic coordinate system is an absolute reference system

On a milling machine, workpieces are normally machined according to a workpiece-based **Cartesian coordinate system** (a rectangular coordinate system named after the French mathematician and philosopher Renatus Cartesius, who lived from 1596 to 1650). The Cartesian coordinate system is based on three coordinate axes designated X, Y and Z which are parallel to the machine guideways.

The figure to the right (Fig. I.2) illustrates the **right-hand rule** for remembering the three axis directions: the middle finger is pointing in the positive direction of the tool axis from the workpiece toward the tool (the Z axis), the thumb is pointing in the positive X direction, and the index finger in the positive Y direction.
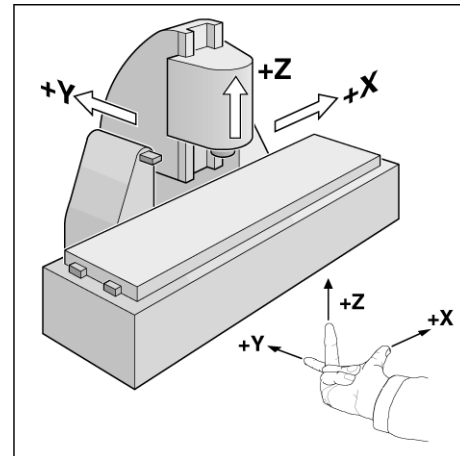
Fig. I.2   Designations and directions of the axes on a milling machine

Setting the Datum

The workpiece drawing (Fig. I.3) 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 the tool radius).

Example:

Drawing with several relative datums (ISO 129 or DIN 406 Part 11, Fig. 171)
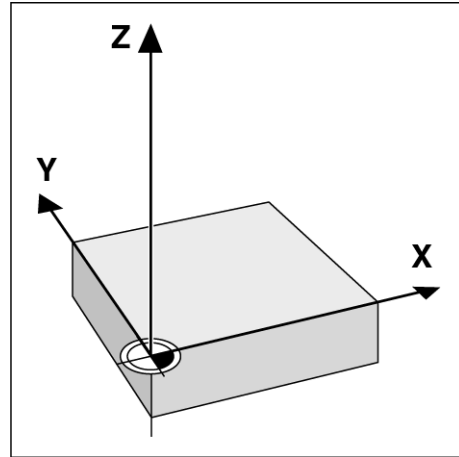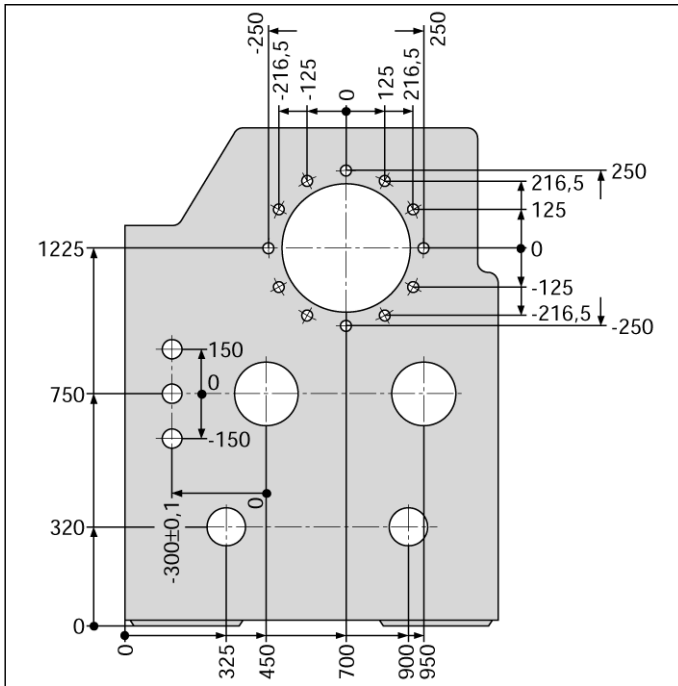
Fig. I.3   The workpiece datum represents the origin of the Cartesian coordinate system

Example: Coordinates of hole **1**:

X =10 mm
Y = 5 mm
Z = 0 mm (hole depth: Z = – 5 mm)
The datum of the Cartesian coordinate system is located 10 mm from hole **1** in the X axis and 5 mm from it in the Y axis. See Fig. I.4.

The KT Edge Finder from **HEIDENHAIN**, together with the POSITIP 880's edge finding functions, facilitates finding and setting datums.
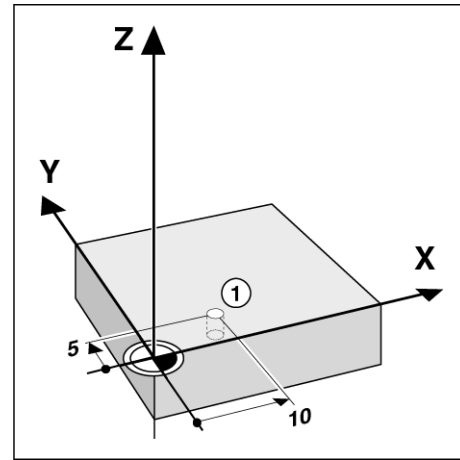
Fig. I.4   Hole 1 defines the coordinatesystem

Nominal Position, Actual Position and Distance-To-Go

The position that the tool is to move to is called the **nominal position** while the position of the tool 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**. See Fig. I.5.

Sign for distance-to-go

The distance-to go has a **positive sign** if the axis direction from the actual towards the nominal position is negative.

The distance-to-go has a **negative sign** if the axis direction from the actual towards the nominal position is positive.
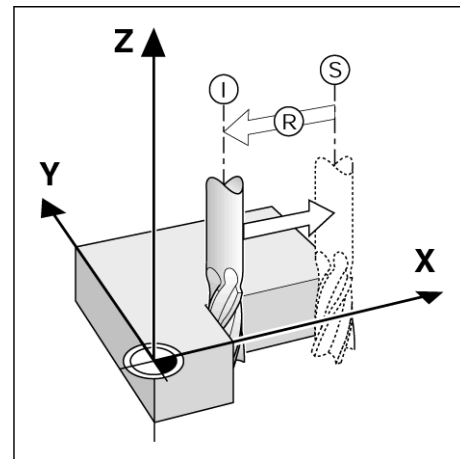
Fig. I.5   Nominal position S, actual position I and distance-to-go R

Absolute Workpiece Positions

Each position on the workpiece is uniquely identified by its absolute coordinates. See Fig. I.6.

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.
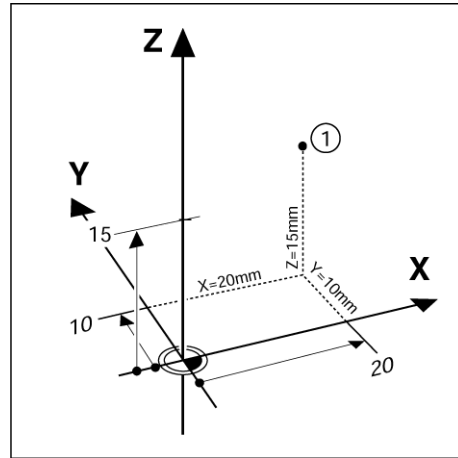
Fig. I.6   Position 1 definition through absolute coordinates

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.7

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.
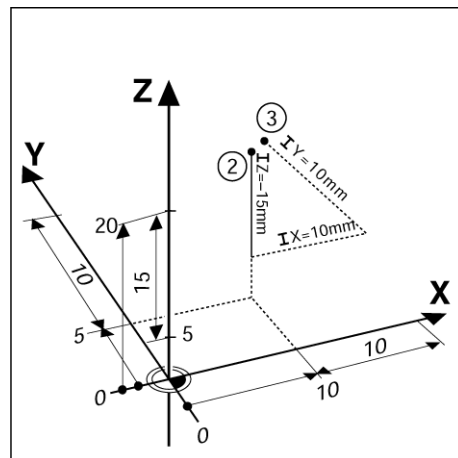
Fig. I.7   Positions 2 and 3 through incremental coordinates

A coordinate list corresponding to this example is useful when working in the operating mode: **PROGRAMMING**. See Fig. I.8
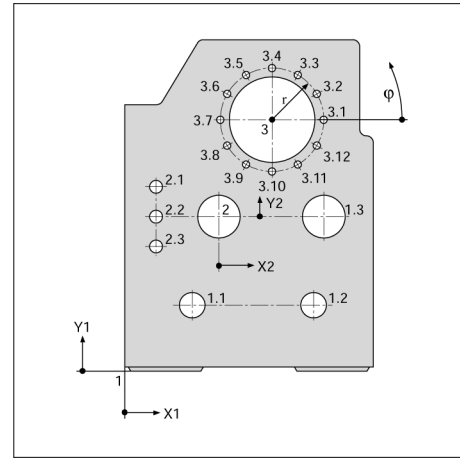
Fig. I.8   Workpiece drawing with coordinate dimensioning (ISO 129 or DIN 406 Part 11, Fig. 179)

Coordinateorigin		Dimensions in mm				
		Coordinates				
	Pos.	X1 X2	Y1 Y2	r	f	d
1	1	0	0			-
1	1.1	325	320			ø 120 H7
1	1.2	900	320			ø 120 H7
1	1.3	950	750			ø 200 H7
1	2	450	750			ø 200 H7
1	3	700	1225			ø 400 H8
2	2.1	-300	150			ø 50 H11
2	2.2	-300	0			ø 50 H11
2	2.3	-300	-150			ø 50 H11
3	3.1			250	0°	ø 26
3	3.2			250	30°	ø 26
3	3.3			250	60°	ø 26
3	3.4			250	90°	ø 26
3	3.5			250	120°	ø 26
3	3.6			250	150°	ø 26
3	3.7			250	180°	ø 26
3	3.8			250	210°	ø 26
3	3.9			250	240°	ø 26
3	3.10			250	270°	ø 26
3	3.11			250	300°	ø 26
3	3.12			250	330°	ø 26

Position Encoders

The **position feedback** encoders convert the movement of the machine axes into electrical signals. The POSITIP 880 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.9.

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 POSITIP 880's reference mark evaluation feature (**REF**).
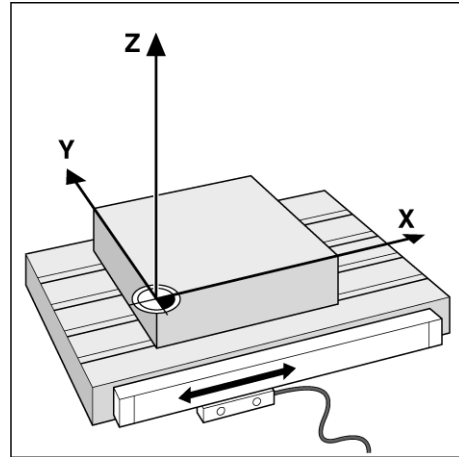
Fig. I.9   Linear position encoder, here for the X axis

Reference Marks

The scales of the position encoders contain one or more reference marks. When reference marks are crossed they can be used to define an absolute position in an incremental system. If power is interrupted this absolute position is lost and the relationship between the reference mark and scale position is lost. The reference marks on the position encoders and the POSITIP 880's reference mark evaluation feature allows the unit to quickly re-establish this relationship again when power is restored. See Fig. I.10.

When a reference mark is crossed over, it generates a signal which identifies that position as the reference point. The POSITIP 880 uses this reference point to restore the relationship between the scale position and the display value which was last defined by setting the datum.

If the position encoders feature **distance-coded** reference marks, there are reference marks uniquely spaced along the length of the scale. Crossing any two reference marks will restore the datum. Each axis need only move a limited distance for linear encoders, and angle for rotary encoders.

Fig. I.10   Linear scales: with distance-coded reference marks (upper illustration) and one reference mark (lower illustration)

> The datum setting cannot be restored from one power cycle to the next if the reference marks were not crossed before the datum was set.

Angle Reference Axis

For angular positions, the following reference axes are defined:

Plane	Angle reference axis
XY	+X
YZ	+Y
ZX	+Z

Positive direction of rotation is counterclockwise if the working plane is viewed in negative tool axis direction. See Fig. I.11.

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
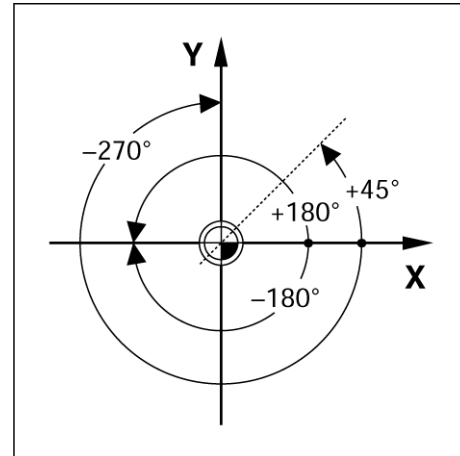
Fig. I.11   Angle and the angle reference axis, e.g. in the X / Y plane

I – 2 Working with POSITIP 880 – First Steps

Power Up

Switch on the power (located on the back). It will take approximately 25 - 30 seconds for system to start after power up. The initial screen will appear **(This screen will only appear the first time you power up)**. Select the language by pressing the **LANGUAGE** soft key.

At this point you have the choice of **MILL** or **TURN.** Select the **MILL** soft key to proceed with milling functions the first time you power up. See Table of Contents for turning functions section. See Fig. I.12.

You can change the application later in INSTALLATION SETUP under COUNTER SETTINGS.

Fig. I.12   Initial screen

Your POSITIP 880 is now ready for operation and is in the operating mode **ACTUAL VALUE**. Axis will show "NO REF".

Before You Start

The POSITIP 880'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 NO REF indicator will flash. See Fig. I.13. After crossing over the reference marks, the indicator will stop flashing and change to REF.

Working without reference mark evaluation

You can also use the POSITIP 880 without crossing over the reference marks. Press the **NO REF** soft key to exit the reference mark evaluation routine and continue in ACTUAL VALUE mode. The NO REF indicator will indicate that reference marks were not crossed over for that axis.

You can still cross over reference marks at a later time. The ENABLE REF soft key will be available from ACTUAL VALUE mode. Press this soft key to activate the reference mark evaluation routine.
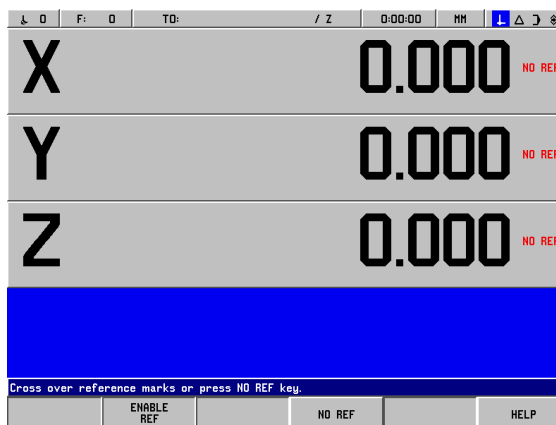
Fig. I.13   Display before choosing **NO REF**

ENABLE REF function

The purpose of the **ENABLE REF** function, is to give the operator the opportunity to either ignore the reference marks as they are crossed over by disabling it, or finding reference marks when necessary by enabling it. When the **ENABLE REF** soft key is pressed, the POSITIP 880 is ready to identify a reference mark. When the **ENABLE REF** soft key is not depressed, POSITIP 880 will ignore all reference marks. When all reference marks have been found, the **ENABLE REF** soft key will disappear.

> If an encoder is setup without reference marks, then the REF indicator will not be displayed.

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 you do **not** cross over the reference marks, POSITIP 880 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).

Turn on power and press any key.

Cross over the reference marks (in any order).

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

Operating Modes

Selecting the **operating mode** determines which functions are available to you.

Available functions	Mode	Key
Position display for workpiece machining; Zero reset; Datum setting – also with edge finder probe	ACTUAL VALUE	
Distance-to-go display; hole patterns; milling and drilling with tool radius compensation	DISTANCE-TO-GO	
Storage of work steps for small-lot production	PROGRAMMING	
Run programs previously created in the **PROGRAMMING** mode	EXECUTE PROGRAM	

You can switch to another operating mode at **any time** by pressing the key for the desired mode.

In the following examples requiring tool usage, refer to chapter I-8 under Job Setup.

On-Screen Operating Instructions (HELP Mode)

The integrated operating instructions provide information and assistance in any situation. See Fig. I.14 & Fig. I.15.

To **call** the operating instructions:

▶ Press the **INFO** soft key.

▶ Press the **HELP** soft key.

▶ Information relevant to the current operation will be displayed.

▶ Use the paging soft 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 paging soft keys to scroll through the index.

▶ Press the **VIEW TOPIC** soft key to select the item you need.

To **leave** the operating instructions:

▶ Press the **EXIT HELP** soft key.

Example: On-screen operating instructions for datum setting with the edge finder (CENTER LINE)

The CENTER LINE function is described in this manual on page 20.

▶ From the **ACTUAL VALUE** mode, press the **PROBE** soft key.

▶ Press the **INFO** soft key.

▶ Press the **HELP** soft key.

▶ To leave the operating instructions: Press the **EXIT HELP** soft key. The screen returns to the screen with the SET DATUM form and DRO display.

Fig. I.14   Index under HELP mode
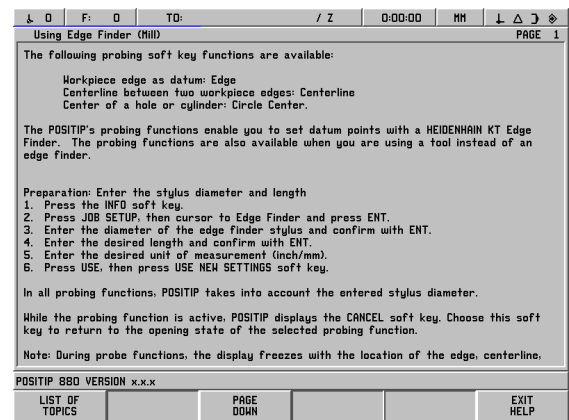
Fig. I.15   On-screen operating instructions for datum setting

Confirming Your Changes

You must confirm your changes by pressing the **ENT** key for them to become effective. The instruction sections of this manual will occasionally give the command "Confirm your changes." This means press the **ENT** key.

Messages

The Message bar messages will change color depending on the type of information it is conveying: Normal messages will appear as gray boxes with black text. Instructional messages will appear as blue boxes with white text. Error messages will appear as red boxes with white text.

Error messages

If an error occurs while you are working with POSITIP 880, the Message bar will turn red and provide an explanation of what caused the error.

To **clear** the error message:

▶ Press the **CE (Clear Entry)** key.

Critical error messages

> ⚠ Critical error messages mean that the operational reliability of the POSITIP 880 has been impaired.

If a critical error occurs, a message box will appear in the middle of the screen:

▶ Take note of the error message displayed on the screen.
▶ Switch off the power to the POSITIP 880.
▶ Attempt to correct the problem with the power off.
▶ If the critical error message recurs, notify your customer service agency.

Selecting the Unit of Measure

Positions can be displayed and entered in millimeters or inches. If you choose inches, INCH will be displayed on the status bar at the top of the screen. See Fig. I.16.

To **change** the unit of measure:

▶ Press the **INFO** soft key.

▶ Press the **INCH/MM** soft key.

▶ The unit of measure can also be set in JOB SETUP. Refer to Job Setup, chapter I-8.
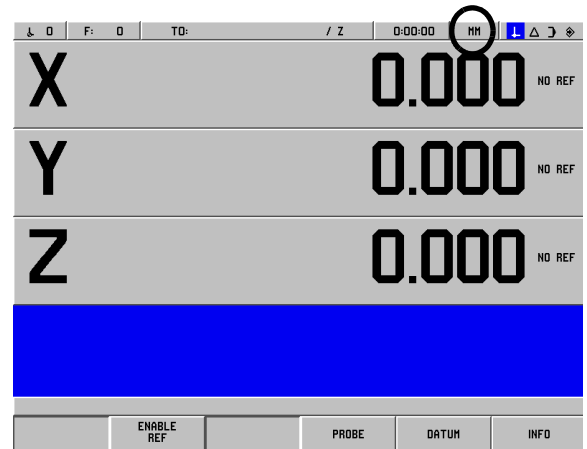
Fig. I.16 The MM indicator

Selecting the Angle Format

Angles – such as for a rotary table – can be displayed and entered either as decimal degrees, degrees/minutes/seconds (DMS) or radian values. Refer to Job Setup, chapter I-8 for instructions on setting angle format.

Tool Table

The POSITIP 880's tool table provides a convenient way to store diameter and length offset information for each of the tools you commonly use. You can enter up to 99 tools.

Before you start workpiece machining, select the tool you are using from the tool table. POSITIP 880 will then take into account the entered diameter and length of the tool.

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.17.

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: $\Delta L < 0$ (–)

Refer to Job Setup for entering a tool into the tool table.
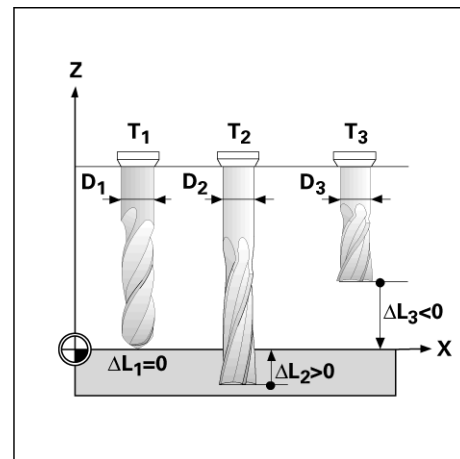
Fig. I.17 Tool length and diameter

Calling the Tool Data

The lengths and diameters of your tools must first be entered into the POSITIP 880's tool table.

Before you start machining, select the tool you are using from the tool table. POSITIP 880 then takes into account the stored tool data when you work with tool compensation (e.g., with hole patterns).

👆 You can also call the tool data with the command **TOOL CALL** in a program.

Tool call

INFO	Press the **INFO** soft key.
JOB SETUP	Then the **JOB SETUP** soft key. (Tool table is highlighted).
(ENT)	Press **ENT**.

TOOL NUMBER

Cursor to the tool you want or enter the tool number directly.

USE	Press **USE**, then **USE NEW SETTINGS**.
USE NEW SETTINGS	

View status bar to verify the proper tool has been called.

I – 3 Actual Value

Datum Setting: Approaching Positions and Entering Actual Values

The easiest way to set datum points is to use the POSITIP 880's probing functions – regardless of whether you probe the workpiece with the **HEIDENHAIN** KT Edge Finder or with a tool. Description of the probing functions: See "Probing Functions for Datum Setting" on page 28

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

The datum table can hold up to 99 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.

Datum settings are performed in the ACTUAL VALUE operating mode and define the relationships between the axis positions and the display values. If necessary, the datum table's values may be changed by entering a value directly.

See Job Setup for setting the datum value directly.

Datum selection (Mill only)

In ACTUAL VALUE mode, the Up/Down arrow keys may be used to select the next or previous datum number.

Tool selection (Turn only)

In ACTUAL VALUE mode, the Up/Down arrow keys may be used to select the next or previous tool number.

Example: Setting a workpiece datum without the probing function. See Fig. I.18 & Fig. I.19.

Working plane: XY

Tool axis: Z

Tool diameter: D = 3 mm

Axis sequence in this example: X - Y - Z

Preparation: Calling the tool data

Select the tool data for the tool which you are using to touch the workpiece.

Preparation: Select the datum

Operating mode: **ACTUAL VALUE**

DATUM	From ACTUAL VALUE mode, press the **DATUM** soft key.
▼	
	Cursor will be in the DATUM NUMBER field.
▼	
(ENT)	Enter the datum number and press **ENT**.
�’	Touch the workpiece at edge **1**.
●	Select the X axis key.

DATUM SETTING X

⊖①.⑤	Enter the position of the tool center (X = – 1.5 mm) and
	transfer the X-coordinate of the datum and press **ENT**.
▼	
�’	Touch the workpiece at edge **2**.
▼	
●	Select the Y axis key.
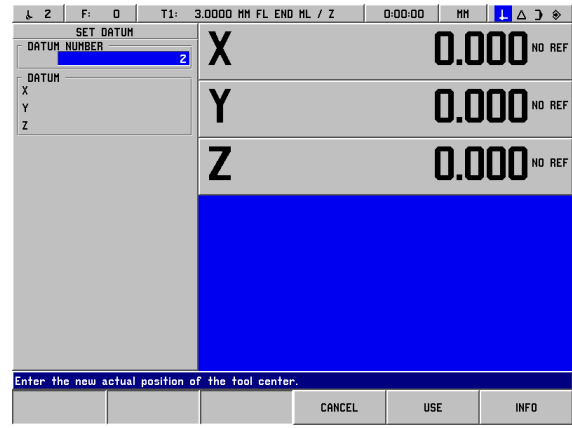
Fig. I.18   SET DATUM form
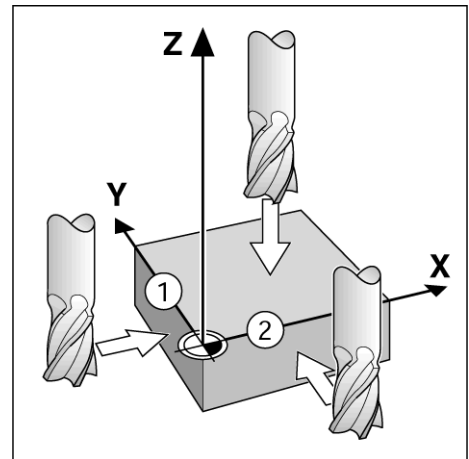
Fig. I.19

DATUM SETTING Y

Enter the position of the tool center (Y = – 1.5 mm) and

transfer the Y-coordinate of the datum and press **ENT**.

Touch the workpiece surface.

Select the Z axis key.

DATUM SETTING Z = + 0

Enter the position of the tool tip (Z = 0 mm) and transfer the Z-coordinate of the datum. Press **USE**.

USE

```
         SET DATUM
 DATUM NUMBER
                        2
 DATUM
 X           -1.500
 Y           -1.500
 Z                 0
```
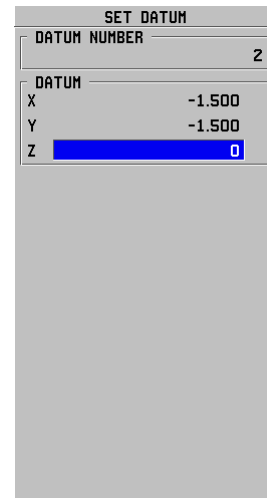
Probing Functions for Datum Setting

It is particularly easy with a **HEIDENHAIN** KT 130 Edge Finder (Fig. I.20).

☞ During probe functions, the display freezes with the location of the edge, centerline, or circle center.

The following probing soft key functions are available:

- Workpiece edge as datum: **EDGE**
- Centerline between two workpiece edges: **CENTER LINE**
- Center of a hole or cylinder: **CIRCLE CENTER**
  With Circle Center, the hole must be in a main plane. The three main planes are formed by the axes X / Y, Y / Z and Z / X.

The POSITIP 880's probing functions enable you to set datum points with a **HEIDENHAIN** KT Edge Finder. The probing functions are also available when you are using a tool instead of an edge finder.

Fig. I.20   The **HEIDENHAIN** KT 130 Edge Finder

Datum setting with the edge finder

Preparation: Enter the stylus diameter and select the datum

▶ Press the **INFO** soft key.

▶ Press **JOB SETUP**, then cursor to EDGE FINDER and press **ENT**.

▶ Enter the diameter of the edge finder stylus and confirm with **ENT**.

▶ Enter the desired length and confirm with **ENT**.

▶ Enter the desired unit of measurement (inch/mm).

▶ Press **USE**, then **USE NEW SETTINGS**.

In all probing functions, POSITIP 880 takes into account the entered stylus diameter.

To abort the probing function

While the probing function is active, POSITIP 880 displays the **CANCEL** soft key. Choose this soft key to return to the opening state of the selected probing function.

Example: Probe workpiece edges, and set the corner as a datum. See Fig. I.21 & Fig. I.22.

Datum axis: X = 0 mm

                          Y = 0 mm

The coordinates of the datum can be set by probing edges or surfaces and capturing them as datums as described on the next page.

Operating mode: **ACTUAL VALUE**

PROBE	Press the **PROBE** soft key.
⬆⬇	Cursor to axis.
EDGE	Press **EDGE** soft key.

PROBE IN X AXIS

Move the edge finder toward the workpiece until the LEDs on the edge finder light up.

Retract the edge finder from the workpiece.

ENTER VALUE FOR X + 0

0	0 is offered as a default value for the coordinate. Enter the desired coordinate for the workpiece edge, for example X = 0 mm and
ENT	set the coordinate as a datum for this workpiece edge. Press **ENT**.
EDGE	Press **EDGE** soft key.

PROBE IN Y AXIS

Move the edge finder toward the workpiece until the LEDs on the edge finder light up.

Retract the edge finder from the workpiece.
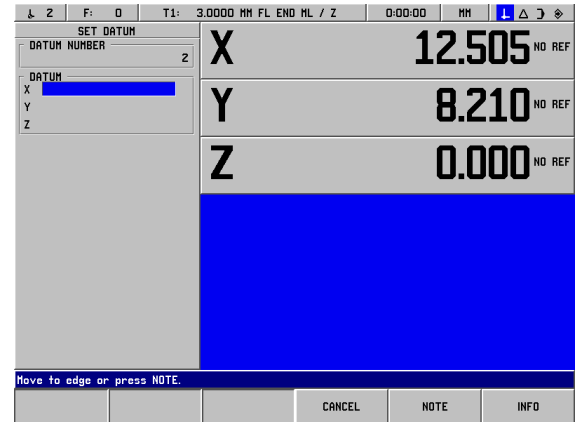
Fig. I.21 Form for setting datum using an edge
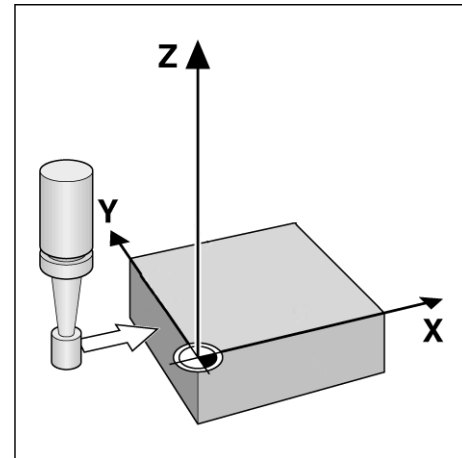
Fig. I.22

ENTER VALUE FOR Y + 0

0

0 is offered as a default value for the coordinate. Enter the desired coordinate for the workpiece edge, for example Y = 0 mm and

ENT

set the coordinate as a datum for this workpiece edge. Press **ENT**.

USE

Press **USE** soft key.

Example: Set centerline between two workpiece edges as datum. See Fig. I.23 & Fig. I.24.

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

The centerline is parallel to the Y axis.

Desired coordinate of the centerline: X =0 mm

Operating mode: **ACTUAL VALUE**

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

PROBE

Press **PROBE**.

⬆⬇

Cursor to the axis for which the coordinate is to be set: X axis.

CENTER LINE

Press **CENTER LINE**.

1. PROBE 1ST EDGE IN X

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

2. PROBE 1ST EDGE IN X

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

Retract the edge finder from the workpiece.
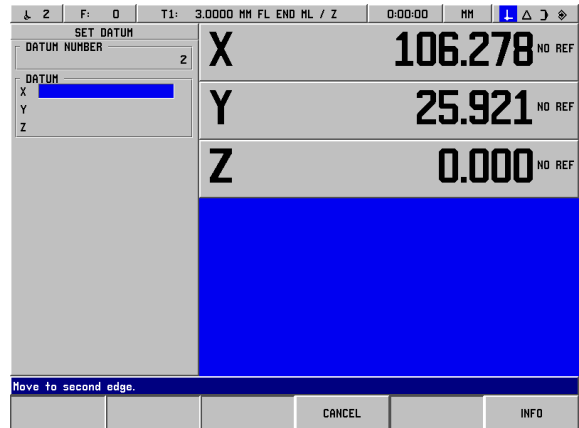
Fig. I.23   Setting centerline between two edges
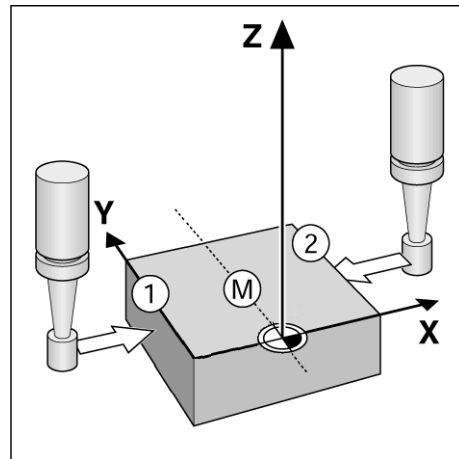
Fig. I.24

0 **ENT**   Enter coordinate (X = 0 mm) and transfer coordinate as datum for the centerline and press **ENT**.

USE   Press **USE** soft key.

Example: Probe the center of a hole with an edge finder and set the datum. Offsetting X-axis datum by 50 mm from center of hole. See Fig. I.25 & Fig. I.26.

Main plane: XY

Edge finder axis: parallel to the Z axis

X coordinate of the circle center: X = 50 mm

Y coordinate of the circle center: Y = 0 mm

Operating mode: **ACTUAL VALUE**

PROBE   Press **PROBE**.

⬆️⬇️   Cursor to the axis for which the coordinate is to be set: X axis.

CIRCLE CENTER   Press **CIRCLE CENTER.**

XY   Select plane (X/Y, Y/Z or Z/X) containing the circle (main plane): Plane XY
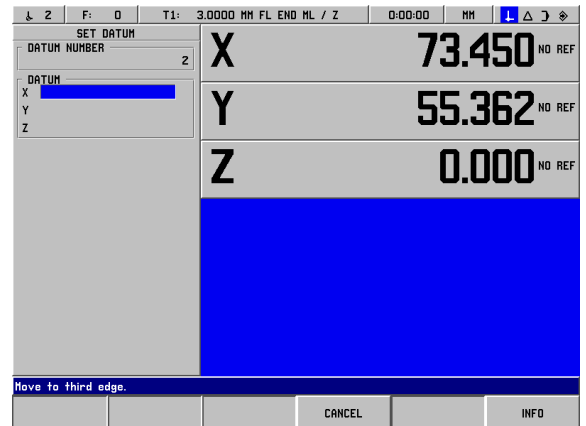
Fig. I.25   Setting the center of a hole
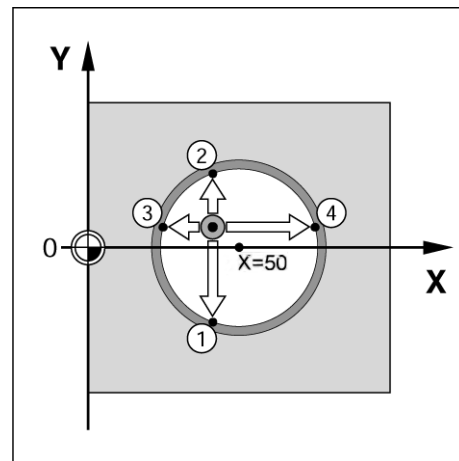
Fig. I.26

PROBE 1ST POINT IN X/Y

Move edge finder towards first point **1** on the circumference until the LEDs on the edge finder light up.

Retract edge finder from bore hole wall.

Probe two additional points on the circumference in the same manner. Further instructions appear on the screen. See Prompt bar for measured diameter.

ENTER CENTER POINT X   X = 50

5 **0**   Enter the first coordinate (X = 50 mm) and

transfer coordinate as datum for the circle center and press **ENT**.

ENTER CENTER POINT Y   Y = 0

0   Accept default entry Y = 0 mm. Press **ENT**. Press **USE.**

Datum Setting with a Tool

Even if you use a tool or non-electrical edge finder to set datum points, you can still use POSITIP 880's probing functions described under the section "Datum setting with the Edge Finder." EDGE, CENTER LINE and CIRCLE CENTER. See Fig. I.27 & Fig. I.28.

Preparation: Enter the tool diameter and select the datum
▶ Press **INFO.**
▶ Press the **JOB SETUP** soft key.
▶ Select the TOOL TABLE form by pressing **ENT**.
▶ Cursor to the tool you will use to set the datum.
▶ Press **USE** and then **USE NEW SETTINGS.**

Example: Probe workpiece edge and set edge as datum
Datum axis: X =0 mm

Tool diameter D = 3 mm

Operating mode: **ACTUAL VALUE**

Fig. I.27   Setting datum using an edge

PROBE	Press **PROBE.**
	Cursor to the axis for which the coordinate is to be set: X axis.
EDGE	Press **EDGE** soft key.
PROBE IN X	
(⊙)	Touch workpiece edge
NOTE	Store the position of the edge by pressing the **NOTE** soft key.
(⊙)	Retract the tool from the workpiece.
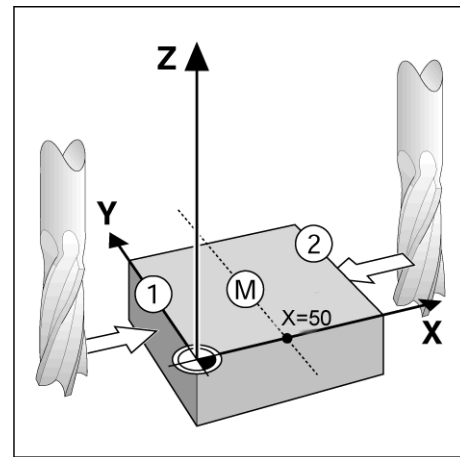
Fig. I.28

ENTER VALUE FOR X

− **1** **.** **5**

Enter coordinate of the tool center

(X = -1.5 mm) and

USE

transfer coordinate as datum for the center line and press **USE** soft key.

I – 4 Distance-To-Go

Displaying and Moving to Positions

Distance-To-Go feature

Although it is often sufficient to have POSITIP 880 display the coordinates of the actual position of the tool, it is usually better to use the **distance-to-go** feature — this enables you to approach to nominal positions simply by traversing to display value zero. Even when working with the distance-to-go feature you can enter coordinates in **absolute** or **incremental dimensions**. Make sure you are in DISTANCE-TO-GO mode.

Graphic positioning aid

When you are traversing to display value zero, POSITIP 880 displays a graphic positioning aid. See Fig. I.29.

POSITIP 880 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.

Refer to the JOB SETUP menu to setup the graphic positioning aid.

Tool Selection

In DISTANCE-TO-GO mode, the up/down arrow keys may be used to select the next or previous tool number

Taking the tool radius into account

POSITIP 880 has a tool radius compensation feature. This allows you to enter workpiece dimensions directly from the drawing. The displayed remaining distance is then automatically lengthened (R+) or shortened (R–) by the value of the tool radius. See Fig. I.30.

Entering tool data

Enter tool data with the **TOOL TABLE** soft key.

▶ Choose **TOOL TABLE** soft key.
▶ Cursor to the desired tool and press **ENT**. The TOOL DESCRIPTION form will appear.
▶ Enter the tool diameter.
▶ Enter the tool length.
▶ Enter the tool unit.
▶ Enter the tool type and press **ENT**.
▶ Press **USE** twice.
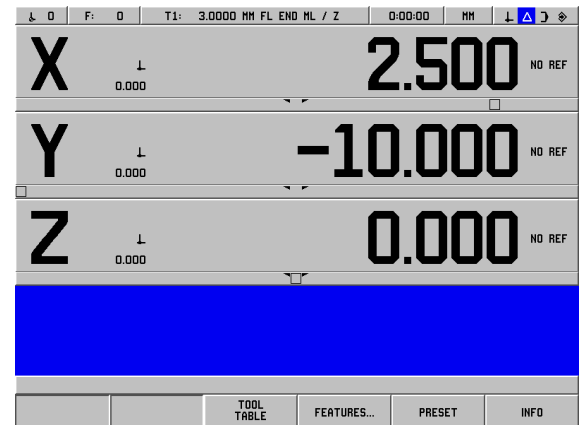
Fig. I.29 Graphic Positioning Aid in use
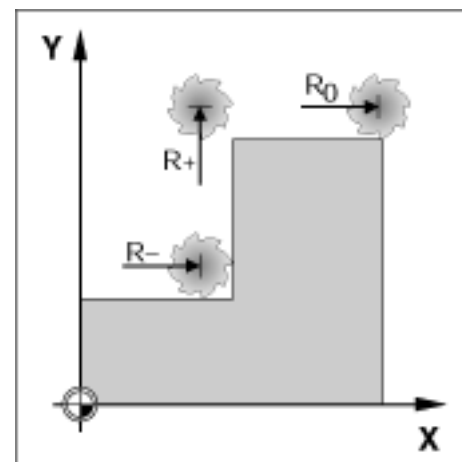
Fig. I.30 Tool radius compensation

Example: Milling a shoulder by traversing to display value zero

The coordinates are entered as absolute dimensions; the datum is the workpiece zero. See Fig. I.31 & Fig. I.32.

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
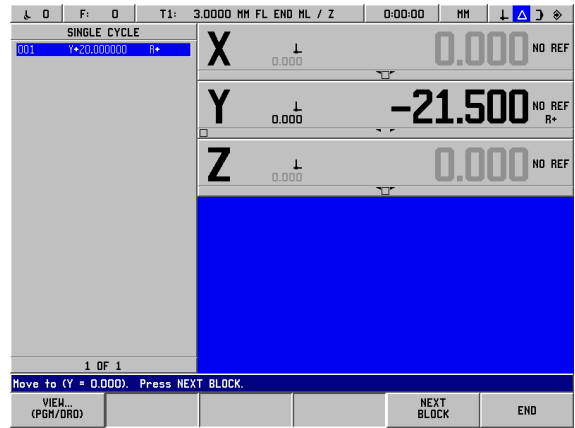
Fig. I.31   Single cycle preset

Operating mode: **DISTANCE-TO-GO**

Preparation:

▶ Enter the tool data.

▶ Pre-position the tool to an appropriate location (such as X = Y = -20 mm).

▶ Move the tool to milling depth.

Select the Y axis key.

NOMINAL POSITION VALUE

2 0   Enter nominal position value for corner point 1:
Y = 20 mm and

TOOL COMP (R+)   select tool radius compensation R + with **TOOL COMP (R+)** soft key.

USE   Transfer the nominal position value by pressing **USE.** The graphic positioning aid for the Y axis is displayed.

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.

Once the axis is moved to zero the axis value is incremented to axis value in the next block.

NEXT BLOCK   Press the **NEXT BLOCK** soft key to advance.
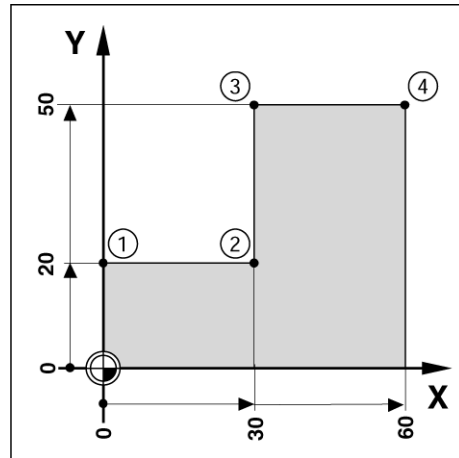
Fig. I.32

⬤   Select the X axis key.

3 **0**   Enter nominal position value for corner point **2**:
Y = +30 mm,

TOOL COMP (R-)   select tool radius compensation R – and press **USE** soft key.

USE   Transfer the nominal position value. The graphic positioning aid for the X axis is displayed.

   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.

Once the axis is moved to zero it will increment to next block.

NEXT BLOCK   Press the **NEXT BLOCK** soft key.

⬤   Select the Y axis key.

5 **0**   Enter nominal position value for corner point **3**:
Y = +50 mm,

TOOL COMP (R+)   select tool radius compensation R + and press **USE** soft key.

USE   Transfer the nominal position value. The graphic positioning aid for the Y axis is displayed.

   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.

Once the axis is moved to zero it will increment to Next Block.

NEXT BLOCK   Press the **NEXT BLOCK** soft key.

● Select the X axis key.

NOMINAL POSITION VALUE

6 **0** Enter nominal position value for corner point **4**:
Y = +60 mm,

TOOL COMP
(R+) select tool radius compensation R + and press **USE**
soft key.

USE

Transfer the nominal position value. The graphic
positioning aid for the X axis is displayed.

⊕ 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.

Once the axis is moved to zero it will increment to
next block.

NEXT
BLOCK Press the **NEXT BLOCK** soft key.

Example: Drilling by traversing to display value zero

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.33 & Fig. I.34.

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

Distance from hole **2** to hole **1**: **I**X = 30 mm / **I**Y = 30 mm

Hole depth: Z = –12 mm

Operating mode: **DISTANCE-TO-GO**

Fig. I.33 Drilling example

NOMINAL POSITION VALUE

⬤ Select an axis key: X axis.

2 **0** Enter nominal position value for hole **1**: X = 20 mm and press **ENT**. Cursor will move to the Y axis field.

NOMINAL POSITION VALUE

2 **0** Enter nominal position value for hole **1**: Y = 20 mm and press **ENT.** Cursor will move to the Z axis field.

NOMINAL POSITION VALUE

– **1** **2** Enter the nominal position value for the hole depth: Z = –12 mm The graphic positioning aid for the Z axis is displayed. Press **USE** soft key.

Pre-position the drill over the first hole. 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 in the tool axis (Z).

Once the axis is moved to zero it will increment to next block.

NEXT BLOCK Press the **NEXT BLOCK** soft key.

⬤ Select an axis key: X axis.

Fig. I.34

NOMINAL POSITION VALUE

3 **0**

I

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

mark your input as an incremental dimension, press **I** hard key.

TOOL COMP (R0)

Select tool radius compensation R0 by pressing **TOOL COMP (R0)** soft key.

USE

Press **USE.** The graphic positioning aid for the X axis is displayed.

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.

Once the axis is moved to zero it will increment to next block.

NEXT BLOCK

Press the **NEXT BLOCK** soft key.

Select the Y axis key.

NOMINAL POSITION VALUE

3 **0**
I

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

mark your input as an incremental dimension, press **I** hard key.

TOOL COMP (R0)

Select tool radius compensation R0 by pressing **TOOL COMP (R0)** soft key.

USE

Press **USE.** The graphic positioning aid for the Y axis is displayed.

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.

Once the axis is moved to zero it will increment to next block.

NEXT BLOCK

Press the **NEXT BLOCK** soft key.

Select the Z axis key.

NOMINAL POSITION VALUE

– **1** **2**

Enter the nominal position value for the hole depth: Z = –12 mm

USE

Press **USE** soft key. The graphic positioning aid for the Z axis is displayed.

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 in the tool axis (Z).

Once the axis is moved to zero the operation is complete.

NEXT BLOCK

Press the **NEXT BLOCK** soft key.

I – 5 Milling Patterns

This chapter describes the hole pattern functions CIRCLE PATTERN, LINEAR PATTERN, RECTANGLE POCKET.

In the operating mode **DISTANCE-TO-GO**, use the soft keys to select the desired hole pattern function or pocket milling, and enter the required data. This data can usually be taken from the workpiece drawing (e.g. hole depth, number of holes, dimensions of the pocket, etc.).

With hole patterns, the POSITIP 880 then calculates the positions of all the holes and displays the pattern graphically on the screen. With pocket milling, it calculates all of the traverse paths for the roughing out of the pocket. The graphic positioning aid appears when you begin execution, enabling you to position simply by traversing to display value zero.

Circle Pattern

Automatic advance function

Once you have moved the table into position and machined to depth, the POSITIP 880 will automatically advance. You can also manually advance by pressing the **NEXT HOLE** soft key.

> POSITIP 880 will not automatically advance if the depth is set to THRU.

Information required:

- Pattern type (full or half)
- Holes (no. 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)
- Step angle (optional: this only applies if creating a circle segment.)
- Depth (the target depth for drilling in the tool axis)

POSITIP 880 calculates the coordinates of the holes which you then move to simply by traversing to display value zero. The graphic positioning aid is available for all moving axes.

A circle pattern is executed as a series of presets to the hole location in the pattern plane and the drill depth.

Functions for drilling

Function	Soft key
Press to go to previous hole.	PREVIOUS HOLE
Press to manually advance to the next hole.	NEXT HOLE
Press to end drilling.	END

Example: Enter data and execute a circle pattern. See Fig. I.35 & Fig. I.36.

Holes (no. of): 4
Coordinates of center: X = 50 mm / Y = 50 mm
Bolt circle radius: 20 mm
Start angle: Angle between X axis and first hole: 30°
Hole depth: Z = – 5 mm

1st step: Enter data

Operating mode: **DISTANCE-TO-GO**

PROGRAMMING FEATURES...   Press **FEATURES** soft key.

CIRCLE PATTERN   Press **CIRCLE PATTERN** soft key.

Begin inputting data.

PATTERN TYPE

FULL CIRCLE   Enter the type of circle pattern (full).

ENT   Press **ENT**.

HOLES

4 ENT   Enter the number of holes (4).

Press **ENT**.
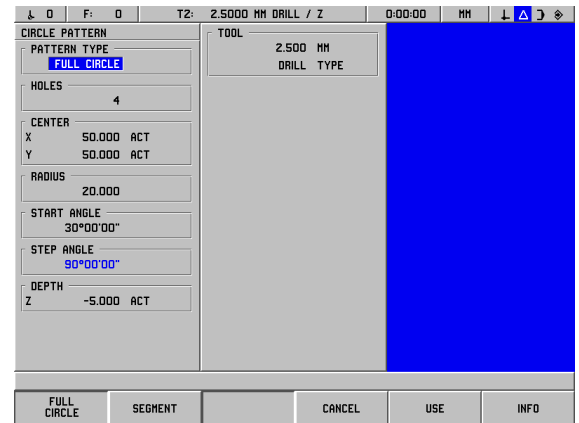
Fig. I.35   CIRCLE PATTERN form
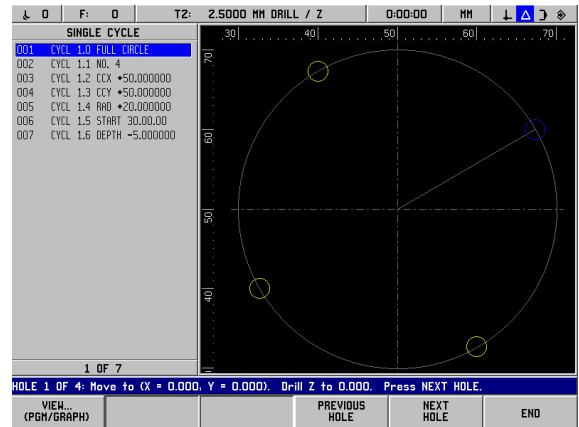
Fig. I.36   Circle pattern graphic

CENTER

5 **0**   Enter the X and Y coordinates of the circle center
(**both** 50).

ENT   Press **ENT**.

RADIUS

2 **0**   Enter the radius of the circle pattern (20).

ENT   Press **ENT**.

START ANGLE

3 **0**   Enter the start angle (30°).
        Press **ENT**.

ENT

DEPTH

–5.0   Enter the depth of each hole (-5.0).

ENT   Press **ENT**.

👆 When in TOOL field enter the TOOL TABLE and select the desired tool.

USE
▶ Press **USE** soft key to begin step.

▶ Pressing the **VIEW ()** soft key will toggle through the different views of the pattern (Graphic, etc.).

2nd step: Drill

Move to hole:
Traverse each coordinate of the working plane to display value zero.

Drill:
Traverse to display value zero in the tool axis. When you have reached required depth, it will automatically advance to the coordinates of the next hole.

After drilling, **retract** the drill in tool axis.

| NEXT HOLE | Press the **NEXT HOLE** soft key. |

Continue to drill the remaining holes in the same way.

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

Linear Pattern

Information required:

- 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)

POSITIP 880 calculates the coordinates of the holes which you then move to simply by traversing to display value zero.

The graphic positioning aid is available for all moving axes.

The graphic enables verification of the hole pattern before you start machining. It is also useful when:

- selecting holes directly
- executing holes separately
- skipping holes

Example: Entering data and executing linear pattern. See Fig. I.37 & Fig. I.38.

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: Z = THRU
Number of rows: 3
Row spacing: 12 mm
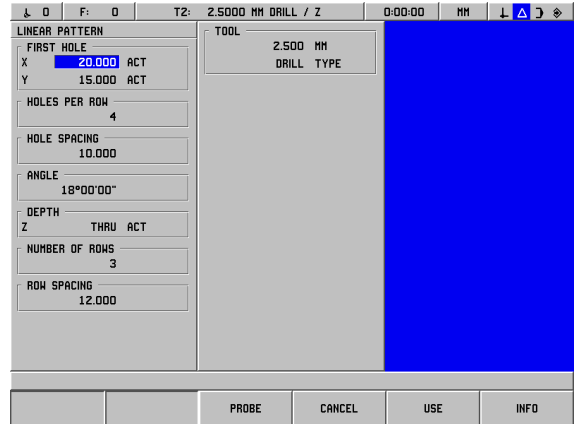
Fig. I.37   LINEAR PATTERN input form

1st step: Enter data

Operating mode: **DISTANCE-TO-GO**

FEATURES...	Press **FEATURES** soft key.
LINEAR PATTERN	Press **LINEAR PATTERN**.
	Begin data entry.

FIRST HOLE X AND Y

2 **0**	Enter the coordinates of first hole (X = 20 mm).
ENT	Press **ENT**.
1 **5**	Y = 15 mm.
ENT	Press **ENT**.
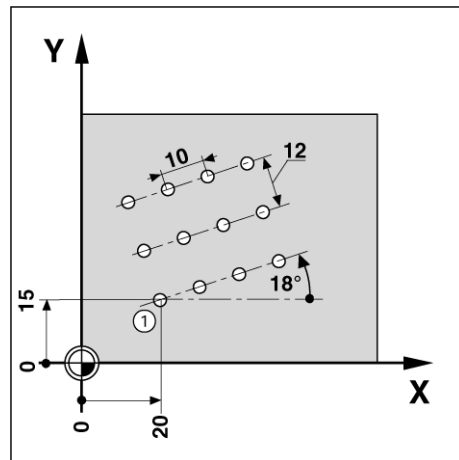
Fig. I.38

HOLES PER ROW

4 **(ENT)**   Enter the number of holes per row (4). Press **ENT**.

HOLE SPACING

1 **0**   Enter the space **between** holes (10).

(ENT)   Press **ENT**.

ANGLE

1 **8**   Enter the angle of rotation (18°).

(ENT)   Press **ENT**.

DEPTH

| THRU | Enter the depth for drilling (THRU).

(ENT)   Press **ENT**.

NUMBER OF ROWS

3   Enter the number of rows (3).

(ENT)   Press **ENT**.

ROW SPACING

1 **2**   Enter the spacing between rows (12).

(ENT)   Press **ENT**.

☞   When in TOOL field enter the TOOL TABLE and select the desired tool.

| USE | Press **USE.**

Press the **VIEW ()** soft key to see graphic.

2nd step: Drill

Start linear hole pattern function.

Move to hole:
Traverse each coordinate of the working plane to 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. |

Continue to drill the remaining holes in the same way.

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

Milling a Rectangle Pocket

In the operating mode **DISTANCE-TO-GO** you can use the POSITIP 880 for milling a rectangle pocket.

The information for rectangle pocket milling can also be written to a machining program as a **cycle**.

Select the cycle with the **RECTANGLE POCKET** soft key, and enter the required data. This data can usually be taken quite easily from the workpiece drawing (e.g. the axis size and the depth of the pockets).

The POSITIP 880 calculates the rough-out paths and offers graphic positioning aid.

Finish allowance
The amount of material to be left for machining on the last pass around the pocket.

Rectangle pocket milling in programs

The POSITIP 880 makes the roughing out of rectangle pockets simple: You just enter the dimensions for the pocket, and the POSITIP 880 calculates the rough-out paths.

Execution of cycle

The execution of the cycle is represented in the figures to the right.

Step 1:
The POSITIP 880 gives the distances-to-go for positioning the tool at the starting position A: First in the tool axis, and then in the machining plane to the center of the pocket.

Step 2:
Roughing out the pocket in accordance with the path indicated in the diagram climb milling (see Step 2). In the working plane the stepover is no greater than the radius of the current tool. The stepover distance of the last pass around the pocket is equal to the finish allowance. The operator chooses the appropriate pecking depth in the tool axis.

Step 3:
This procedure repeats itself until the entered depth B is reached.

Input into cycle for **RECTANGLE POCKET**

■ Starting position
  (enter absolute value, referenced to datum)
■ End depth
  (enter absolute value, referenced to datum)
■ Center in X
  Center of the pocket in the main axis of the working plane.
■ Center in Y
  Center of the pocket in the secondary axis of the working plane.
■ Size length in X
  Length of the pocket in the direction of the main axis.
■ Size length in Y
  Length of the pocket in the direction of the secondary axis.
■ Direction
  (counter-clockwise or clockwise)
■ Finish allowance
  Finishing allowance in the working plane.
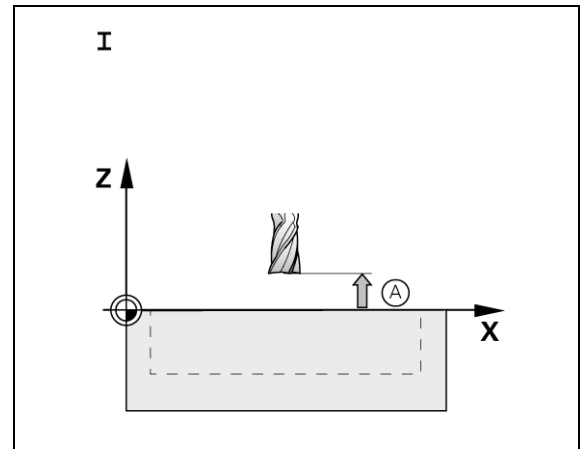
Step 1 in rectangle pocket
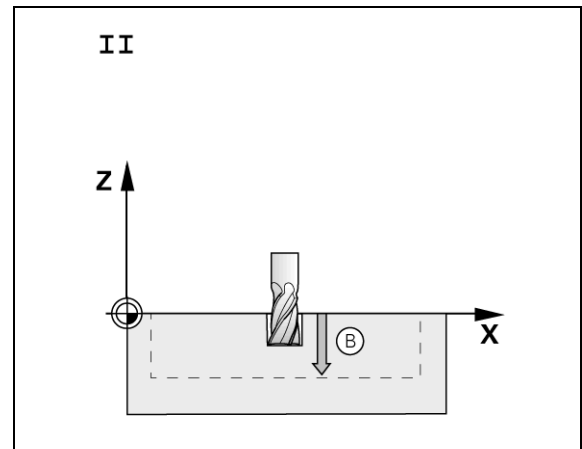
Step 2 in rectangle pocket
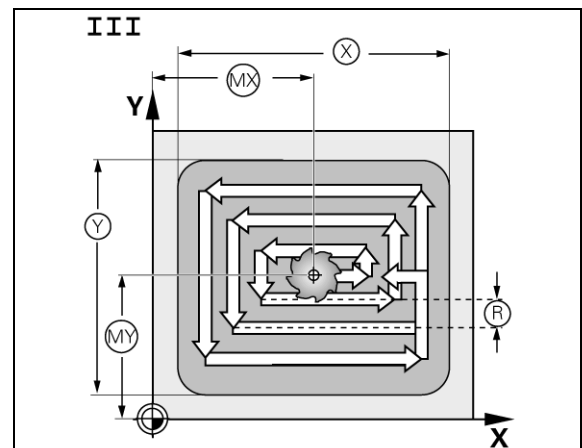
Step 3 in rectangle pocket

Programming example: Mill rectangle pocket. See Fig. I.39 & Fig. I.40.

Starting position: 2 mm
Milling depth: –20 mm
Pocket center in X: 50 mm
Pocket center in Y: 40 mm
Side length in X: 80 mm
Side length in Y: 60 mm
Direction: CCW
Finishing allowance: 0.5 mm

Example: Entering rectangle pocket data into a program

Operating mode: **DISTANCE-TO-GO**
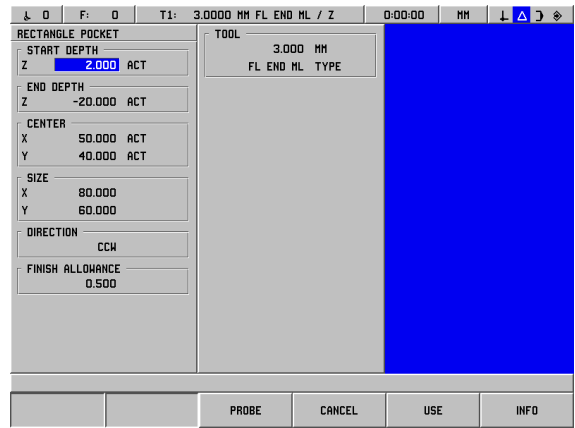
Fig. I.39 Rectangle pocket programming example

Fig. I.40 Rectangle pocket programming graphic

FEATURES...	Press **FEATURES**.

RECTANGLE POCKET	Press **RECTANGLE POCKET.**

START DEPTH

2 (ENT)	Enter the **start depth** (2 mm). Press **ENT**.

END DEPTH

– 2 0	Enter the **end depth** (- 20 mm). Press **ENT**.

CENTER

5 0 / 4 0	Enter the X and Y dimensions for the pocket center. Press **ENT**.

SIZE

8 0 / 6 0	Enter the X and Y dimensions for the side. Press **ENT**.

DIRECTION

CCW	Enter the direction of rotation (counter-clockwise). Press **ENT**.

FINISH ALLOWANCE

0 **.** **5**   Enter the amount of finish allowance (0.5).

Press **ENT**.

When you cursor to the TOOL field, press **TOOL TABLE** soft key. Select tool and press **USE.** Tool will enter into the TOOL field.

2nd step: Mill a rectangle pocket

END

After you have entered all of the required data, start the RECTANGLE POCKET cycle and position the axes by traversing to zero. The pecking depth in the tool axis does not have to be preset. End the cycle after the pocket has been milled pressing **END**.

I – 6 Programming POSITIP 880

Program Capabilities

Operating mode: **PROGRAMMING**

This section describes the programming capabilities of the POSITIP 880. The programs can be edited in program memory and executed in the EXECUTE PROGRAM mode. The operator can load, save or delete a program from internal storage in FILE OPERATIONS. The CLEAR function clears the current program in memory. See Fig. I.41.

Function	Soft key
Allows access to all programming features.	PROGRAMMING FEATURES...
Allows the operator to load, save, delete or clear a program.	FILE OPERATIONS...
Allows the operator access to all block functions.	BLOCK FUNCTIONS...
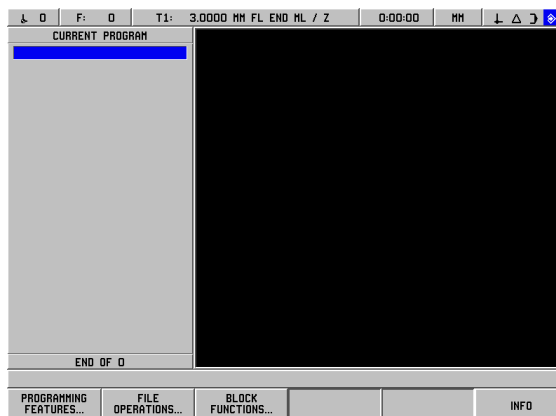
Fig. I.41 Program function screen

Editing and moving through a program

- Press **Up/Down Arrow** keys to move the program block highlight cursor to the next or previous block. The highlight cursor will wrap around the first and last blocks in the program.
- To move to a specific block number in the current program, enter a number and press **ENT**.
- To insert a new step in the current program, press **PROGRAM FEATURES** soft key and select the desired block. The block is inserted above the block currently highlighted.
- To edit the block currently highlighted, press the **ENT** key.
- To delete the block currently highlighted from the program, press the **CE** key.

Programming Features

The PROGRAMMING mode has a **PROGRAMMING FEATURES** soft key which shows all the blocks that can go into a program. Below is a list of **PROGRAMMING FEATURES** soft keys.

Operating mode: **PROGRAMMING**

Function	Soft key
Opens the TOOL CALL block which is used to select a specific tool number from the tool table at run-time.	TOOL CALL
DATUM CALL is used to select a datum at run-time.	DATUM CALL
PRESET is used to locate a target position on an axis. The PRESET form allows for entering one or more axis values. Each axis value will be entered as a separate preset block.	PRESET
Opens CIRCLE PATTERN form. Refer to the Milling Patterns chapter for performing circle patterns.	CIRCLE PATTERN
Opens LINEAR PATTERN form. Refer to the Milling Patterns chapter for performing linear patterns.	LINEAR PATTERN
Opens the RECTANGLE POCKET form. Refer to the Milling Patterns chapter for performing rectangle pockets.	RECTANGLE POCKET
Opens the LABEL NUMBER form to mark the beginning and the end of a subprogram	LABEL NUMBER
LABEL CALL (REPEAT) is used to call a subprogram 1 or more times. It allows the subprogram to be offset a specified amount on each iteration.	LABEL CALL (REPEAT)
LABEL CALL (Rotate) allows blocks within a subprogram to be rotated a specified amount on each iteration.	LABEL CALL (ROTATE)
LABEL CALL (MIRROR) allows blocks within a subprogram to be inverted over an axis to represent a mirror image of that subprogram.	LABEL CALL (MIRROR)
POSITION DRILL soft key opens the program form for position drill jobs.	POSITION / DRILL
LINE soft key opens the program form for milling a line.	LINE

Function	Soft key
ARC soft key opens the program form for milling an arc.	ARC
BLEND soft key opens the program form and NORMAL ARC, INVERTED ARC and CLOSED CONTOUR soft keys for performing a blend.	BLEND
CHAMFER soft key opens the program form and the CLOSED CONTOUR soft key to create a chamfer.	CHAMFER
MORE soft key allows the operator to toggle through the different soft keys rows.	MORE...

☞ Some programming features provided by the Positip 880, such as Arcs, Blends and Chamfers, are not easily machined on a manually operated machine. These tool paths require two axes with simultaneously controlled motion. The Positip 880's Contour View function provides a graphical method for the operator to approximate these tool paths. See "Contour View" on page 81for details regarding this function.

Tool Call

The TOOL CALL is used to select a specific tool number from the tool table at run-time, for instance during a program. The block specifies a tool number and an optional tool axis. See Fig. I.42.

Calling a tool

Refer to Job Setup for how to enter the length and diameter of your tools in the tool table.

The TOOL CALL command automatically pulls the tool length and diameter from the tool table.

You may define the tool axis for the program run in the TOOL CALL block.

Tool axis setting

A tool axis is selected to define the machining plane. Tool radius compensation is applied to the axes forming the machining plane. Tool length compensation is applied along the tool axis.

For example, a horizontal boring machine may have machining features such as hole patterns located in the YZ plane. The boring operations are performed in the X axis. For this configuration, the tool axis would be set to the X axis.

■ Press the **TOOL AXIS** soft key.

■ Select the tool axis with soft key.

■ Press USE.

> When you execute a tool call block with a new tool axis, this tool axis will remain in effect until you select another from the tool table or another tool call block.

To perform a tool call

Operating mode: **PROGRAMMING**

| PROGRAMMING FEATURES... | Press the **PROGRAMMING FEATURES** soft key. |

| TOOL CALL | Select the **TOOL CALL** soft key. The cursor defaults to the NUMBER field. |

TOOL NUMBER

| ENT | The tool number defaults to the previous tool block's number. Enter the desired value by using the numeric key pad or press the **TOOL TABLE** soft key. (The SIZE and TOOL TYPE fields are read only.) Press **ENT**. |

Fig. I.42 TOOL CALL form

TOOL AXIS (OPTIONAL)

(ENT)   Entering the tool axis (such as Z) is optional. It is used to set the tool axis at run-time. An axis may be selected by using the axis (**X**, **Y**, **Z**) soft keys. Press **ENT**. The program contains the tool call block TOOL CALL 1 Z.

(CE)   Choose **CE** to clear any tool axis selection.

[USE]   Press **USE.**

Datum Call

The DATUM CALL block is used to select a datum at run-time. See Fig. I.43.

Datum call

The POSITIP 880 can store up to 99 datums in a datum table. You can call a datum point from the datum table during a program run by simply pressing the **DATUM CALL** soft key and entering the block DATUM XX. This automatically calls the datum point entered for **XX** during program run.

Operating mode: **PROGRAMMING**

[PROGRAMMING FEATURES...]   Press the **PROGRAMMING FEATURES** soft key.

[DATUM CALL]   Select the **DATUM CALL** soft key. Call a datum point from the table.

Fig. I.43 DATUM CALL form

DATUM NUMBER

[5]   Enter the datum number (such as 5). Press **USE.** Input range: 0 to 99.

[USE]   DATUM #5 will appear within the block.

Presets

The operator uses the PRESET function to establish a nominal position on an axis to which to move. Use the PRESET form to enter the desired preset value. The PRESET form allows for entering one or more axis presets. However, multiple axis presets are entered into the program as separate preset blocks. See Fig. I.44.

Entering a Preset

An axis preset involves entering a target value, selecting a preset type (ACT or INC), and selecting a tool offset.

☞ If the **PRESET** soft key is used, the cursor defaults to the first axis field.

Operating mode: **PROGRAMMING**

PROGRAMMING FEATURES...	Press the **PROGRAMMING FEATURES** soft key.
PRESET	Select the **PRESET** soft key or one of the axis keys.

NOMINAL VALUE

I	Enter a nominal value using the numeric keys.
	Press the Incremental hard key to toggle between ACT and INC preset types. For an ACT preset, the nominal value is relative to the current datum (at run-time). For an INC preset, the nominal value is relative to the previous nominal location.

TOOL OFFSET

TOOL COMP (R0)	Press the **TOOL COMP (R0)** soft key to toggle through the R+ (add radius), R- (subtract radius) or R0 (no compensation) choices. Enter the value and
USE	Press **USE.**
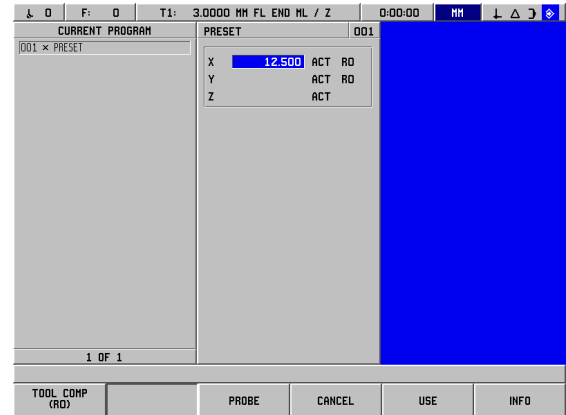
Fig. I.44 PRESET form

Hole Patterns and Rectangle Patterns

See the Milling Patterns section for programming information on circle patterns, linear patterns and rectangle patterns.

Subprograms

Subprograms

A subprogram is a subsection of blocks within a program which can be called from another block within the program. The subprogram blocks are programmed once, but may be run multiple times. A subprogram can also be mirrored. See Fig. I.45.

Nesting subprogram

Subprograms can also be "nested." For example, a subprogram can in turn call another subprogram.
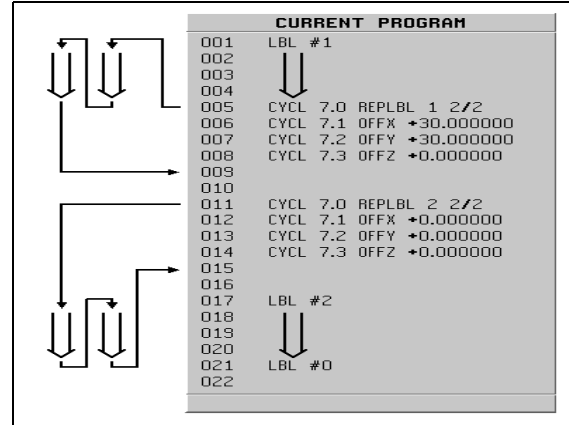
Fig. I.45   Subprogram

Labels

Labels are visual markers within a program that designate the beginning and end of a subprogram (abbreviated in the program to LBL.)

Example: Insert label for subprogram. See Fig. I.46.

Operating mode: **PROGRAMMING**

PROGRAMMING FEATURES...	Select **PROGRAMMING FEATURES**.
MORE...	Select the **MORE** soft key to advance to the second soft key row.
LABEL NUMBER	Insert a label (LBL) for a subprogram.

LABEL NUMBER

1 **USE**	Enter a label number (1). Press **USE.** The current block now contains the label LBL #1.
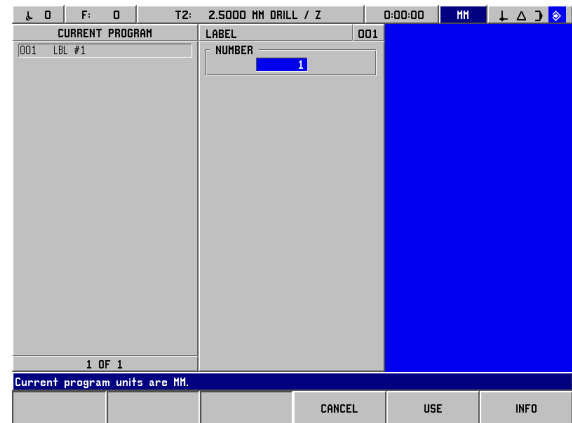
Fig. I.46   LABEL form

Label Number

Label numbers 1 to 99 identify the beginning of a subprogram.

Label 0

Label 0 is used only to identify the end of a subprogram. An end label is optional. If not used, the subprogram will end either at the block from which the subprogram was called or after the last block in the program.

Label Call

In the program, subprograms and program sections are called with the command LBLCALL, such as LABEL CALL (REPEAT), LABEL CALL (ROTATE), LABEL CALL (MIRROR).

The command LBL CALL 0 is not allowed.

After a LBL CALL block is inserted into the program, POSITIP 880 executes the called subprogram.

LABEL CALL REPEAT

This form is used to call a subprogram 1 or more times. It allows the subprogram to be offset a specified amount on each iteration. See Fig. I.47.

LABEL CALL ROTATE

This form allows blocks within a subprogram to be rotated a specified amount on each iteration.

LABEL CALL (MIRROR)

This form allows blocks within a subprogram to be inverted over an axis to represent a mirror image of that subprogram.

The beginning of a subprogram (or a program section repeat) is now marked with the label. Enter the program blocks for the subprogram after the LBL block.

Label 0 is used **only** to identify the **end** of a subprogram.

Fig. I.47 LABEL CALL REPEAT form

Example: Enter a subprogram named `LBL CALL REPEAT`.

PROGRAMMING FEATURES...	Press the **PROGRAMMING FEATURES** soft key again and advance to the next soft key row by pressing the **MORE** soft key.
MORE...	
LABEL CALL (REPEAT)	Call the label by pressing the **LABEL CALL (REPEAT)** soft key.
1	Enter the label number of the subprogram to be called (1).
3 0	Enter X offset of 30. Press **ENT**. Enter Y offset of 30. Press **ENT**.
1	Enter the number of repeats (1).
USE	Press **USE.**

After the LBL CALL block in the operating mode **EXECUTE PROGRAM**, the POSITIP 880 executes those blocks in the subprogram that are located between the LBL block with the called number and the next block containing LABEL 0. All subprograms should start with a label number (1 to 99) and end with a label 0.

Position Drill

Point X1: 20 mm
Point Y1: 20 mm
Depth: Z = -12.00 ABS

POSITION DRILL function will move your table to the position you want based upon your desired X and Y values.

In the operating mode **PROGRAMMING** you can use the POSITIP 880 for a position drill.

Select the cycle with the **POSITION/DRILL** soft key, and enter the required data. See Fig. I.48. This data can usually be taken quite easily from the workpiece drawing.

Example: Enter data for a position drill

Operating mode: **PROGRAMMING**

PROGRAMMING FEATURES...	Press the **PROGRAMMING FEATURES** soft key.
MORE...	Press **MORE** soft key.
POSITION / DRILL	Press the **POSITION/DRILL** soft key.
	Begin data entry.
X	
2 **0**	Enter the point for X. Press **ENT**.
Y	
2 **0**	Enter the point for Y. Press **ENT**.
DEPTH	
— **1** **2**	Enter the cut depth. Press **ENT**.

Fig. I.48 POSITION/DRILL form

Milling a Line

In the operating mode **PROGRAMMING** you can use the POSITIP 880 for milling a line.

Lines are defined by their "FROM" point (the point where they begin) and "TO" point (the point where they end).

There are two ways you can program a line:

- With four coordinates(X1, Y1) (X2, Y2)
- With three of the coordinates above (X1, Y1) or (X2 or Y2) and an angle.

Choose a method based upon the information available from your print.

Select the cycle with the **LINE** soft key, and enter the required data. See Fig. I.49. This data can usually be taken quite easily from the workpiece drawing.

The POSITIP 880 calculates the tool path and offers graphic positioning aid.

Example: Enter data and mill a line

From point X1: 4 mm
From point Y1: 2 mm
To point X2: 2 mm
To point Y2: 2 mm
Depth: Z= 4.00
Tool offset: Center

1st step: Enter data for a line

Operating mode: **PROGRAMMING**

Fig. I.49   MILL LINE form

| PROGRAMMING FEATURES... | Press the **PROGRAMMING FEATURES** soft key, then the **MORE** soft key. |
| MORE... | |

| LINE | Press the **LINE** soft key. |

Beginn data entry.

FROM X1

4   Enter the from point for X. Press **ENT**.

FROM Y1

2   Enter the from point for Y. Press **ENT**.

TO X2

2 Enter the to point for X. Press **ENT**.

TO Y2

2 Enter the to point for Y. Press **ENT**.

DEPTH

4 Enter the cut depth. Press **ENT**.

TOOL OFFSET

CENTER   Enter the tool offset. Press **ENT**.

USE   Press **USE.**

Milling an Arc

In the operating mode **PROGRAMMING** you can use the POSITIP 880 for milling an arc. See Fig. I.50.

There are two ways you can program an arc:

■ With a from point, to point and a radius
■ With a from point, to point and a center point

Choose a method based upon the information available from your print.

Fig. I.50 MILL ARC form

Major and Minor Arcs - A Major arc has a sweep angle greater than 180 degrees. A Minor arc has a sweep angle less than 180 degrees. See Fig. I.51 as a graphic example of the two types of angles.

Example: Enter data and mill an arc

From point X1: 6 mm
From point Y1: 3 mm
To point X2: 3 mm
To point Y2: 6 mm
Depth: Thru
Radius: 2.24, < 180°
Direction: CW
Tool offset: Center

1st step: Enter data for an arc

Operating mode: **PROGRAMMING**
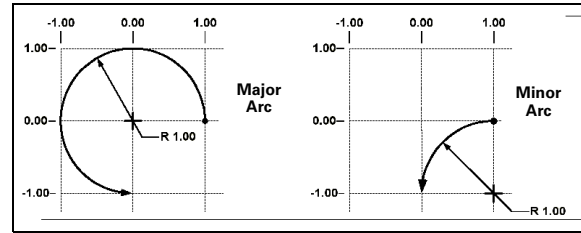
Fig. I.51

| PROGRAMMING FEATURES... | Press the **PROGRAMMING FEATURES** soft key. |

| ARC | Press **ARC** soft key. |
| | Beginn data entry |

FROM X1

| 6 | Enter the from point for X. Press **ENT**. |

FROM Y1

| 3 | Enter the from point for Y. Press **ENT**. |

TO X2

| 3 | Enter the to point for X. Press **ENT**. |

TO Y2

| 6 | Enter the to point for Y. Press **ENT**. |

CUT DEPTH

| THRU | Enter the cut depth. Press **ENT**. |

RADIUS

2 **.** **2** **4**

Enter the size of the radius.

Press **MINOR ARC** soft key. Press **ENT**.

DIRECTION

| CW |

Enter the arc direction. Press **ENT**.

TOOL OFFSET

| CENTER |

| USE |

Enter the tool offset. Press **ENT**.

Press **USE.**

Blend

A blend is a semi-circular fillet which connects two adjacent blocks within a program. The two blocks may be two lines, two arcs, or a line and an arc. The POSITIP 880 calculates the location of the arc to transition smoothly from the previous block to the next. The two blocks do not have to intersect or touch; however, if they do not touch, the blend arc must be large enough to connect them. The blend arc can be specified as "normal" or "inverted". The blend step cannot be executed as a single cycle operation. See Fig. I.53 as a graphic example of different types of blends.

A blend usually connects the previous block in the program with the next block to form a continuous path. It is also possible to have the blend connect the previous block to the first block within the current continuous path.

To close the path using a blend, enter the blend's radius and press the **CLOSE CONTOUR** soft key. See Fig. I.52.
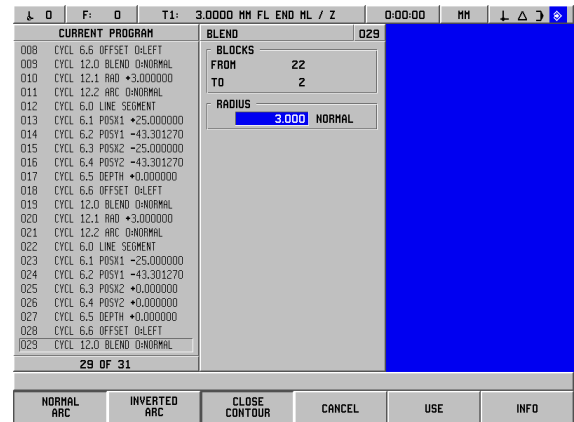
Fig. I.52 Input parameters for a blend

Operating mode: **PROGRAMMING**

PROGRAMMING FEATURES...	Press the **PROGRAMMING FEATURES** soft key.

MORE...	Press the **MORE** soft key twice.

BLEND	Press the **BLEND** soft key.

The "FROM" and "TO" fields are read only. They are automatically filled when the BLEND form is opened.

The "TO" field changes with the **CLOSE CONTOUR** soft key state.

BLEND ARC RADIUS

Enter the blend arc radius by using the numeric keys.

BLEND TYPE

INVERTED ARC	Enter type of radius (normal, inverted).

CLOSE CONTOUR

CLOSE CONTOUR	Press **CLOSE CONTOUR** to set the first step in the continuous path.

Pressing the CLOSE CONTOUR soft key causes the blend to connect the first block in the current path instead of the next block. The block number shown in the "TO" field will change.

When the desired data has been entered, press **USE.**
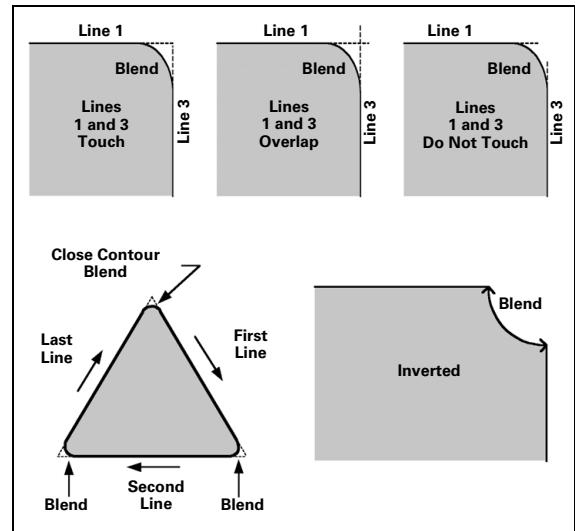
Fig. I.53 Examples of how blend can be used

Chamfer

A chamfer is an angled cut which connects two adjacent line blocks within a program. The POSITIP 880 calculates the location of the chamfer. The two lines do not have to intersect or touch, however, if they do touch, the lengths are calculated from their intersection points. See Fig. I.55 as a graphic example of a chamfer.

A chamfer usually connects the previous block in the program with the next block to form a continuous path. It is also possible to have the chamfer connect the previous block to the first block within the current continuous path. This "closes" the path.

Fig. I.54 Input parameters for a chamfer

Intersecting lines – You can insert a chamfer between two intersecting lines that are adjacent in the program sequence. See Fig. I.54.

A chamfer step can be specified by:

▶ Length 1 and length 2, the angle is blank.

▶ Length 1 or length 2 and the angle.

> 👆 The axis lines must show they are currently at or will come to a point of connection in order to create a chamfer.

Operating mode: **PROGRAMMING**

|	
PROGRAMMING FEATURES...	Press the **PROGRAMMING FEATURES** soft key.
MORE...	Press the **MORE** soft key twice.
CHAMFER	Press the **CHAMFER** soft key. The "FROM" and "TO" fields are read only. They are automatically filled when the CHAMFER form is opened.
CLOSE CONTOUR	To close the path using a chamfer, enter the chamfer's data and press the **CLOSE CONTOUR** soft key.

Fig. I.55 Example of a chamfer

LENGTH 1

Enter the distance from the point of intersection on the line in the "FROM" step. Use the numeric keys to enter the distance.

LENGTH 2

Enter the distance from the point of intersection on the line in the "TO" step.

ANGLE

Enter the angle measured from the line along the X axis and sweeps toward the adjacent line. Use the numeric keys to enter the angle.

CLOSE CONTOUR

| CLOSE CONTOUR |

Pressing the **CLOSE CONTOUR** soft key causes the chamfer to connect the first block in the current path instead of the next block. The block number shown in the "TO" field will change.

| USE |

When the desired data has been entered, press **USE.**

File Operation Soft Keys

Function	Soft key
Load from internal storage or to current program memory	LOAD
Save current program in memory to internal storage	SAVE
Delete program from internal storage	DELETE
Clear current program in memory	CLEAR
Select a directory for internal storage	SELECT DIRECTORY
Create a name for a new directory	CREATE DIRECTORY
Delete a directory from internal storage	DELETE DIRECTORY
Allows the operator to transfer a program from an external device into POSITIP 880's program memory.	IMPORT PROGRAM
Allows the operator to transfer a program from POSITIP 880 to an external device (PC or printer).	EXPORT PROGRAM

Loading, saving, deleting & clearing a program

POSITIP 880 allows you to load programs from internal storage to current program memory.

To load or delete a program:

FILE OPERATIONS...	Press the **FILE OPERATIONS** soft key.
LOAD	Press the **LOAD** soft key.
LOAD	Choose the desired program and press **LOAD.** Confirm with **YES.** Follow the same procedure for deleting a program, except press **DELETE** soft key.

To save a program:

FILE OPERATIONS...	Press the **FILE OPERATIONS** soft key.
SAVE	Press **SAVE** soft key.
	To spell the name of the new program, use the **LEFT** and **RIGHT** soft keys and the Up/Down arrow keys to move through the selection of alphabetized characters.
SELECT CHARACTER	Choose desired characters, by pressing the SELECT CHARACTER soft key to spell the name of the new program.
	To enter numerals in the name, press the numeric keypad.
	Use CE key to backspace.
SAVE	When you have completed spelling the new program name, select the **SAVE** soft key.

To clear a program:

FILE OPERATIONS...	Press the **FILE OPERATIONS** soft key.
CLEAR	Press CLEAR soft key. Confirm with **Yes.**

Directories

POSITIP 880 allows the operator to create directories in which to maintain and store part programs. Press the **MORE** soft key to access these functions.

Function	Soft key
Select a directory	**SELECT DIRECTORY**
Create a name for a new directory	**CREATE DIRECTORY**
Delete a directory	**DELETE DIRECTORY**

Selecting a directory

To select a directory:

| FILE OPERATIONS... | Press the **FILE OPERATIONS** soft key. |

| MORE... | Press **MORE** soft key. |

| SELECT DIRECTORY | Press the **SELECT DIRECTORY** soft key. |

| SELECT | Choose the desired directory and press **SELECT.** |

Creating a directory

To create a directory:

| FILE OPERATIONS... | Press the **FILE OPERATIONS** soft key. |

| MORE... | Press **MORE** soft key. |

CREATE DIRECTORY	Press **CREATE DIRECTORY** soft key.
	Spell the name of the new directory, using the **LEFT** and **RIGHT** soft keys and the Up/Down arrow keys to move through the selection of alphabetized characters.
	To enter numerals to the name use the numeric keypad

| SELECT CHARACTER | Choose desired characters, by pressing the SELECT CHARACTER soft key |
| | Use CE key to backspace. |

| CREATE | When you have completed spelling the new directory name, select the **CREATE** soft key. |

Deleting a directory

To delete a directory:

FILE OPERATIONS...	Press the **FILE OPERATIONS** soft key.
MORE...	Press **MORE** soft key.
DELETE DIRECTORY	Press the **DELETE** DIRECTORY soft key.
DELETE	Choose the desired directory and press **DELETE**. Confirm with **Yes.**

Importing a Program

The RS-232-C interface on the rear panel allows you to utilize a device such as a PC for external data storage.

☞ Pin layout, wiring and connections for the data interface see Chapter II-4 Data Interface.

☞ If the device is not properly connected you will receive an error message.

Example: Importing a program into POSITIP 880

Operating mode: **PROGRAMMING**

FILE OPERATIONS...	Press the **FILE OPERATIONS** soft key.
MORE...	Press the **MORE** soft key twice.

IMPORT PROGRAM

IMPORT PROGRAM	Press the **IMPORT PROGRAM** soft key.
	Setup the PC host terminal program to send the program file as ASCII text data.

If you are importing programs into POSITIP 880 from a PC, the PC must **send** the programs.

POSITIP 880 will load the new program into program memory only. You will have to save it to internal storage if necessary. The external device (PC) must be connected through the serial port.

Exporting a Program

Example: Exporting a program from POSITIP 880

Setup the PC host terminal program to receive the ASCII text data and to save the data to a file.

FILE OPERATIONS...　Press **FILE OPERATIONS**

MORE...　Press **MORE** (twice)

EXPORT PROGRAM　Press **EXPORT PROGRAM**

If you are transferring programs from POSITP 880 to a PC, the PC must be ready to receive the program before pressing **EXPORT PROGRAM.**

Block Function Soft Keys

A block is a single step or a series of steps that make up a program.

Operating mode: **PROGRAMMING**

Press the **BLOCK FUNCTIONS** soft key.

Function	Soft key
This feature will "explode" a program block into several more detailed blocks. Applies to hole patterns and REPEAT, ROTATE and MIRROR blocks.	EXPLODE
Change depth or offset for a range of blocks.	CHANGE BLOCKS
Delete a range of blocks from current program.	DELETE BLOCKS
Copy a range of blocks within the current program.	COPY BLOCKS
Move a range of blocks within the current program.	MOVE BLOCKS
Reverse order of blocks in a path and the "from" and "to" points in each block.	REVERSE PATH
Adjusts the end point of current block and start point of the next block to the intersection point of the two features.	TRIM NEXT

Function	Soft key
Adjusts the start point of current block and end point of the previous block to the intersection point of the two features.	TRIM PREVIOUS
Adjusts the end point of the last block in the path and the start point of the first block in the path to the intersection point of the two features	TRIM PATH

Explode

This feature will "explode" a program block into several more detailed blocks. You can explode hole functions and CALL LABEL REPEAT, ROTATE and MIRROR. The step that you exploded will be replaced with the individual lines and arcs (CALL LABEL REPEAT, ROTATE and MIRROR) or positions (hole patterns) that made up the original block.

Highlight the block you want to explode.

BLOCK FUNCTIONS...   Press the **BLOCK FUNCTIONS** soft key.

EXPLODE   Press the **EXPLODE** soft key.

Changing program blocks. See Fig. I.56.

Operating mode: **PROGRAMMING**

| CHANGE BLOCKS | From within BLOCK FUNCTIONS, press the **CHANGE BLOCKS** soft key. |

Select the range of blocks you want to change (FIRST and LAST). Press **ENT** after each selection.

| THRU | Enter the Z axis depth manually using either the numeric keypad or select the **THRU** soft key. Press **ENT**. |

| USE | Enter the tool offset: **LEFT**, **CENTER** or **RIGHT**. Press **USE.** This applies to lines and arcs within the range only. |

Fig. I.56 Changing a program block

Deleting, copying and moving program blocks

You can delete, copy or move a range of blocks.

When a block is deleted, POSITIP 880 automatically renumbers the remaining blocks. The block after the deleted block then becomes the current block.

Example: Delete program blocks

Operating mode: **PROGRAMMING**

| DELETE BLOCKS | From within BLOCK FUNCTIONS, press the **DELETE BLOCKS** soft key. |

| (ENT) | Select the range of blocks you want to delete (FIRST and LAST). Press **ENT** after each selection. |

| USE | Press **USE**. |

Trim blocks

You can trim, or connect, two adjacent line blocks using the trim feature. The trim feature connects the two blocks by extending each line to the point where they intersect.

TRIM NEXT	Move the cursor to the line block to be trimmed. Press the **TRIM NEXT** soft key. The end point of the current line and the start point of the next line will be set to the point where both lines intersect.
TRIM PREVIOUS	Move the cursor to the line block to be trimmed. Press the TRIM PREVIOUS soft key. The start point of the current line and the end point of the previous line will be set to the point where both lines intersect.
TRIM PATH	Move the cursor to the line block to be trimmed. Press the TRIM PATH soft key. The end point of the last line and the start point of the first line in the path will be set to the point where both lines intersect.

I – 7 Executing a Program

Preparing to Run a Program

When you are ready to run a program, you must be in the **EXECUTE PROGRAM** mode. Press the **EXECUTE PROGRAM** hard key, then the **FILE OPERATIONS** soft key, press the **LOAD** soft key, scroll to the desired program, then press the **LOAD** soft key again. The current program block is displayed at the top of the screen.

Black to gray feature

This feature is an on-screen visual aid. The axes that need to be moved into position will be shown in black in the axes bar. When you get into position they will switch to gray. Note: If your depth axis is set to THRU, the depth axis will not switch to gray.

Operating mode: **PROGRAMMING**

You will be given a choice of how you want to run the program: Single block or automatic.

■ **Single block**
When you have moved the axis to the displayed position, call the next block with the **NEXT BLOCK** soft key. It is recommended that you use **SINGLE BLOCK** when running a program for the first time.

■ **Automatic**
Here the display automatically shows the next program block as soon as you have moved to the displayed position. Press **AUTOMATIC** when you are sure the program contains no errors and you want to run it quickly. When all displayed axes have switched to gray (indicating that they are in position) the program advances to the next block or hole.

Single block example

Operating mode: **EXECUTE PROGRAM**

SINGLE BLOCK	Select **SINGLE BLOCK**. The program block and the graphic positioning aid will appear (if currently in use).

⊕	Position by traversing to display value zero.

NEXT BLOCK	Call the next program block by pressing **NEXT BLOCK**.

Continue positioning and calling blocks with **NEXT BLOCK** until machining is complete.
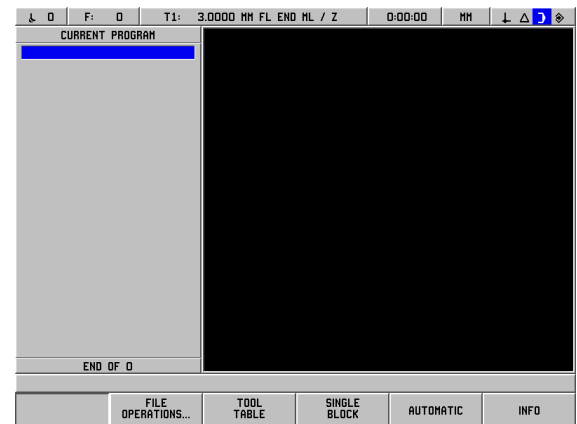
Fig. I.57   1st EXECUTE PROGRAM screen

Automatic example:

Operating mode: **EXECUTE PROGRAM**

| AUTOMATIC | Select **AUTOMATIC.** |

Position by traversing to display value zero.

The display automatically shows the next program block as soon as you have moved to the displayed position. The positioning aid automatically switches to the coordinate axis of the new block.

| NEXT BLOCK | Call the next program block by pressing **NEXT BLOCK**. |

Function	Soft key/Key
Start with the block before the current block (this key is available prior to running a program).	⬆
Start with the block after the current block (this key is available prior to running a program).	⬇
Enter into the tool data (this key is available prior to running a program).	TOOL TABLE
Used in either SINGLE BLOCK or AUTOMATIC mode. Used only during execution of hole patterns. Press to advance to the next hole in a hole pattern.	NEXT HOLE
Used in either SINGLE BLOCK or AUTOMATIC mode. Used only during execution of hole patterns. Press to step back to the previous hole in a hole pattern.	PREVIOUS HOLE
Used in either SINGLE BLOCK or AUTOMATIC mode. Used only during execution of pocket patterns. Press to advance to the next step in a pocket pattern.	NEXT STEP
Used in SINGLE BLOCK or AUTOMATIC mode for the operator to advance manually to the next block within a program.	NEXT BLOCK
Press to abort the AUTOMATIC or SINGLE BLOCK mode and return to the previous screen.	END

Program views

The POSITIP 880 allows the operator to see several different views of the current operation. After choosing **SINGLE BLOCK** or **AUTOMATIC** to begin an operation, the soft key on the far left will offer several viewing options starting with **VIEW (PGM/DRO)**. Toggle through to choose other view options.

Operating mode will automatically switch to: **EXECUTE PROGRAM**

Function	View
Shows all axis coordinates and each input parameter in the current operation.	VIEW... (PGM/DRO)
Shows enlarged view of axis display coordinates.	VIEW... (DRO 1)
Shows the graphical layout of the current machining parameters.	VIEW... (PGM/GRAPH)
Shows the necessary feedback for precise positioning or for following a contour along the machining plane.	VIEW... (CONTOUR)

Contour View

The contour view is an optional display screen available when executing a program. The view will only appear if turned on. Refer to section I-8 Job Setup for instructions. See Fig. I.58 & Fig. I.59.

Graphic displays

The contouring view provides the necessary feedback for positioning precisely or for following a contour along the machine plane.

The screen consists of a main view showing the tool path and hole locations, a smaller window showing the full part, and another smaller DRO window showing current position.

☞ Please note that this display only allows the operator to approximately machine along a complex path. The approximation of the cut to the actual programmed path is solely a function of the skill and patience of the operator.

Main view

The main view is used for finer positioning. It's scaling is such that one pixel on the display is equal to the finer of the X and Y axis display resolutions. This allows scale motion of 1 encoder pulse to be visually discernible.

A tool icon is fixed in the center of the view. The tool path is displayed relative to the tool. As the table is moved, the tool path moves relative to the tool only.

The tool icon is not displayed to scale. It represents the center of the tool only.

The tool path is offset from the programmed contour by the tool radius and tolerance band. The tool's path is relative to the tool selected.

☞ Contour is turned on and off in JOB SETUP.

Contours

For lines and arcs the tool path is shown. The tool path includes any offset specified in the program.

On either side of the tool path is a tolerance band. The tolerance band parameter is setup in JOB SETUP. The gap between the tool path and the tolerance band represents the allowable deviation from the tool path. The inside tolerance band is shown in red. The outside tolerance band is shown in green.
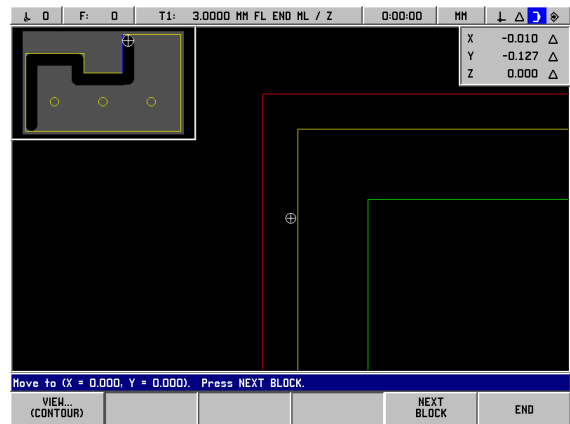
Fig. I.58   Test program in contour view
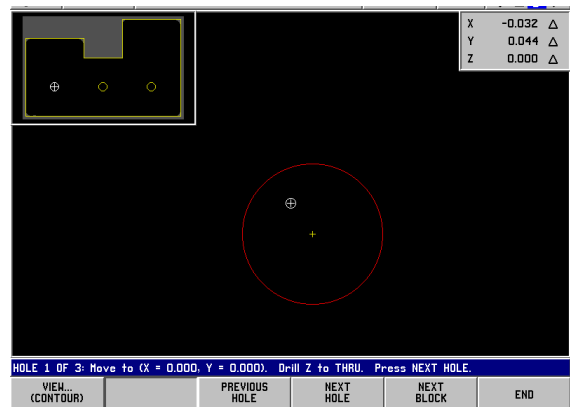
Fig. I.59   Test program in contour view

Positions and hole patterns

For positions and hole patterns, the position or center of the hole is shown as a cross-hair.

A circle around the position/hole location represents the tolerance band. When the center of the tool icon is within the circle, the cutting is within the tolerance band.

Full part window

The full part window provides for very coarse positioning. It shows the tool relative to the entire part. The scaling is such that the entire part will fit within the window with a small buffer all around.

The part profile is fixed. The tool icon is shown to scale using the current tool's radius. As the table moves, the tool icon moves relative to the part.

The shaded area represents the stock material. It is shown slightly larger than the part's extents. As the tool icon moves, the shading below the tool changes to background gray. This material removal allows the shading to be used as an indication of the material remaining.

The material removal mechanism functions only while the contouring view is shown. If the tool is moved while another view is selected, the affected material will not be removed in the contour view.

DRO window

The contour view provides the necessary feedback for positioning precisely or for following a contour along the XY plane.

The DRO window shows the distance-to-go positions for X, Y and Z axis. The DRO window allows viewing the position without switching to another view.

I – 8 INFO Screen

JOB SETUP Menu

The JOB SETUP menu is accessed by pressing the **INFO** soft key the **JOB SETUP** soft key and then **ENT**. See Fig. I.60. When in the JOB SETUP menu, the following soft keys will be available:

■ **INSTALLATION SETUP**
Press to begin accessing the INSTALLATION SETUP parameters.

■ **IMPORT/EXPORT** Press to begin importing or exporting the operating parameters.

■ **CANCEL CHANGES** Will cancel all changes and exit from JOB SETUP back to the previous mode.

■ **USE NEW SETTINGS** Will save any changes to the configuration file before exiting.

■ **HELP**
Will open on-line help.

General field/form navigation guide

■ Press **Up/Down arrow** keys to move between fields.

■ Use keypad to enter numeric values within each field.

■ Press **ENT** to confirm the entry within a field.

■ Press **USE** when form is completed and save settings to the form.

■ Press **USE NEW SETTINGS** to return to the INFO screen.

To access the JOB SETUP menu:

| INFO | Press the **INFO** soft key. |

| JOB SETUP | Press the **JOB SETUP** soft key.

The cursor will default to the first selection on the menu: TOOL TABLE |
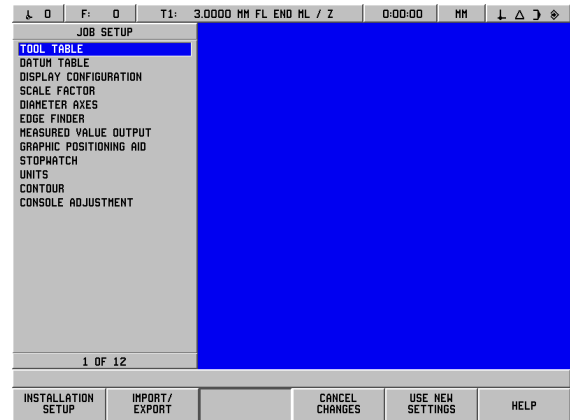
Fig. I.60   JOB SETUP screen in milling

Tool table (Milling only)

The tool table will support up to 99 tools containing information on the diameter, length and type. The length offset may be entered as a known value or the POSITIP 880 may determine the offset automatically. See Fig. I.61.

The following soft keys are available:

Function	Soft key
Press to enter the tool offset length. Only available in TOOL LENGTH field.	TEACH OFFSET
This will open the TOOL TYPES form for selection. Only available in TYPE field.	TOOL TYPES
This key will sort all entered tools by type (i.e., flat end mills with flat end mills, etc.).	SORT BY TYPE
This key will allow the operator to select between tool axes (X,Y,Z).	SELECT TOOL AXIS
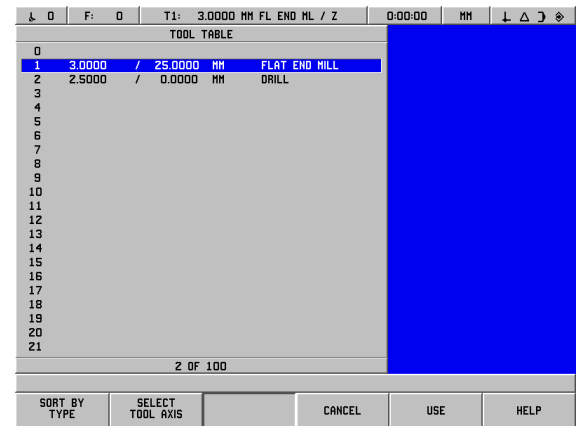
Fig. I.61 Tool table in milling

Tool Table Usage

Example: Entering the tool length and diameter into the tool table. See Fig. I.62 & Fig. I.63.

Tool diameter 3.00

Tool length: 25.000

Tool unit: mm

Tool type: flat end mill

> ☞ It is also possible to have POSITIP 880 determine an offset. See alternative example below.

INFO **JOB SETUP**	Press the **INFO** soft key, then the **JOB SETUP** soft key.
	The cursor will default to the TOOL TABLE field.
(ENT)	Press **ENT**.

TOOL TABLE

⬇ (ENT)	Cursor to the tool you want to define or enter the tool number. Press **ENT**.

TOOL DIAMETER

3	Enter the tool diameter (3.0) and
(ENT)	press **ENT**.
	Cursor will automatically move to the LENGTH field.

Fig. I.62 Tool length and diameter entry

TOOL LENGTH

2 **5**

ENT

If a tool presetter is used to measure the length of a tool, enter the tool length (25 mm) and press **ENT**.

It is also possible to have POSITIP 880 determine an offset. This method involves touching the tip of each tool to a common reference surface. This allows POSITIP 880 to determine the difference between the length of each tool.

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

TEACH OFFSET

Press the **TEACH OFFSET** soft key. POSITIP 880 will calculate an offset relative to this surface. Press **ENT**.

ENT

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 set 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 the reference surface and set the datum to 0.

Cursor will automatically move to the UNIT field.

TOOL UNIT

MM

Enter the tool unit (inch/mm) and

ENT

press **ENT.**

Cursor will automatically move to the TYPE field.

Fig. I.63 TOOL TYPE field

TOOL TYPE

| TOOL TYPES | If you want to set the type, press **TOOL TYPES** soft key. Select from list and press **ENT**. |

| USE | Press **USE** soft key twice, then the **USE NEW SETTINGS** soft key once. |

| USE | |

| USE NEW SETTINGS | |

☝ The tool table may be protected to prevent changes from being made to the tool information. Refer to II-2 Installation Setup.

Tool axis setting

A tool axis is selected to define the machining plane. Tool radius compensation is applied to the axes forming the machining plane. Tool length compensation is applied along the tool axis.

For example, a horizontal boring machine may have machining features such as hole patterns located in the YZ plane. The boring operations are performed in the X-axis. For this configuration, the tool axis would be set to the X-axis.

▶ Press the **TOOL AXIS** soft key.
▶ Select the axis by pressing one of the soft keys.
▶ Press **USE**, then **USE NEW SETTINGS** soft key.

☝ Tool selection and tool axis selection take effect immediately. Pressing the **CANCEL CHANGES** soft key does not restore the previous settings.

Datum table (milling only)

The datum table shows the absolute locations of the datums relative to the reference mark. Normally the datums are set using the datum and probe functions in actual value mode. It is also possible to set the datum values directly if necessary. See Fig. I.64.

▶ Press the arrow keys to cursor to the datum you need and press **ENT**. The datum description form will appear.

▶ Use numeric keys to enter the desired values and press **ENT** for each axis. The values are measured from the reference marks.

▶ When you have finished entering values, press the USE soft key.

Resetting the datum table:

▶ Press the **RESET ALL DATUMS** soft key to reset all axis values in the datum table.

☞ Datum number 0 may be protected to prevent it from being changed. Refer to II-2 Installation Setup.

Fig. I.64 Datum table in milling

Display configuration

The DISPLAY CONFIGURATION form is where the operator determines which axes are displayed and in what order. The DRO display may have 1 or 2 independent configurations.

▶ Press the **NEXT AXIS** soft key to toggle through the available axes.

▶ Press the **CE** key to turn off the axis display.

▶ The form is closed by pressing the **CANCEL** or **USE** soft keys.

▶ The factory default configuration for DRO 1 is 1st = X, 2nd = Y, and 3rd = Z. The 4th, 5th, and 6th displays are OFF. The default for DRO2 is all displays OFF.

▶ When more than 1 configuration is available, the **DRO VIEW** soft key is used to toggle between DRO1 and DRO2. The current setting is shown on the soft key. On power up, the DRO display defaults to DRO1.

Scale factor

The scale factor may be used to scale the part up or down from the programmed size. All displayed values are multiplied by the scale factor. Tool radius offsetting is not affected. A scale factor of 1.0 is used to create a part with the exact size as dimensioned on the print. See Fig. I.65.

▶ The numeric keys are used to enter a number greater than zero. The range is 0.1000 to 10.000.

▶ The scale factor settings will be retained on a power cycle. First-time power-up default setting is OFF with a value of 1.0000.

▶ When the scale factor is a value other than 1, the scaling symbol is shown on the axis display.

Fig. I.65 Scale factor

Diameter axes

Select DIAMETER AXES to set the axis values to either diameter or radius values depending on the requirements of the drawing. .

▶ Cursor to DIAMETER AXES and press **ENT**.

▶ The cursor will be in the X axis field. (The field will default to RADIUS). Depending on the parameter you need for that axis either press the **DIAMETER** soft key or press **ENT** to leave as RADIUS and advance to the next axis field.

▶ Press **USE**, then **USE NEW SETTINGS**.

Radius/Diameter Display

POSITIP 880 can display either the radius or the diameter for you. When the diameter is being displayed, the diameter symbol (Ø) is shown next to the position value.

Edge finder (milling only)

The diameter and length offset of the edge finder are set in this form. Both values are in the units indicated in the form.

The numeric keys are used to enter values of diameter and length. The diameter must be greater than zero. The length is a sign value (negative or positive).

Soft keys are used to select the units for the edge finder dimensions. The entered diameter and length values are not converted when the units are toggled.

The edge finder values will be retained on a power cycle.

Measured value output

With the measured value output feature, the current axis display positions and probe measurement values can be sent over the serial and /or parallel ports. Output of the current display positions is activated via an external hardware signal (requires optional Auxiliary Machine Interface) or via a command (Ctrl B) sent to POSITIP 880 over the serial port.

The Measured Value Output form is used to set data output during probing operations to On or Off. It is also used to set the display freeze option.

◼ Data Output Probing - This may be set to either On or Off. When On, the measurement data is output when the probe operation is completed.

◼ Display Freeze - Set to either:
  ◼ Off - The display is not stopped during measured value output.
  ◼ Concurrent - The display is stopped during measured value output and remains stopped while the switching input is active.
  ◼ Freeze - The display is stopped, but is updated by every measured value output.

Refer to chapter II-5 Measured Value Output for information on the format of the output data.

Graphic positioning aid

The GRAPHIC POSITIONING AID form is used to configure the bar graph that is shown below the axes' DRO display in DISTANCE-TO-GO mode. Each axis has its own range. See Fig. I.66.

▶ Press the **ON** soft key to enable or simply begin entering values using the numeric keys. Enter default value of 5 mm.

▶ Only axes that have been setup in the AXIS CONFIGURATION will appear in the form. Graphic positioning aid will be retained on a power cycle.

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 when the **START** soft key is pressed and stops when **STOP** is pressed.

It will be reset to 0:00:00 when **RESET** is pressed. Resetting will stop the watch if it is running.

The elapsed time field shows the total accumulated time from each interval.

All stop watch functions are applied immediately. When running, both the status bar and stop watch and the elapsed time field are updated in real-time.
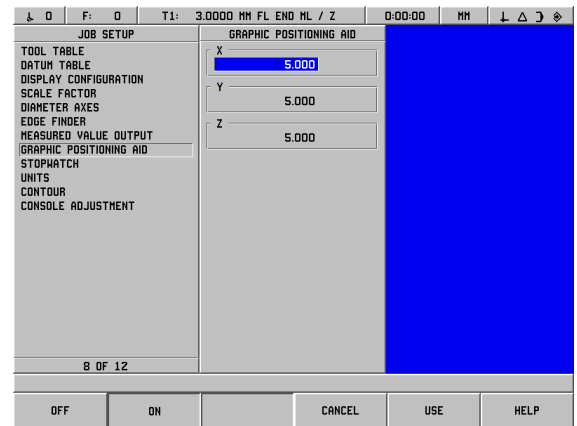
Fig. I.66 Graphic positioning aid

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 DIMENSIONS field. Choose between **INCH** or **MM** soft keys.

- Decimal Degrees, Radians or Degrees/Minutes/Seconds (DMS) - The ANGLE FORMAT field affects how angles are displayed and entered in forms. Choose between these three soft keys **DECIMAL DEGREES**, **RADIANS** or **DMS**.

- CW/CCW - Direction fields in canned cycle forms default to the direction clockwise or counter-clockwise selected in the DIRECTION field.

Contour

The CONTOUR form is used to enable or disable the feature and to set the tolerance bands.

▶ Press the ON soft key to enable the feature. The contour view will be available when executing a program.

▶ Press the OFF soft key to disable the feature.

▶ Enter a value for the inside tolerance. The inside tolerance is shown in red on the screen.

▶ Enter a value for the outside tolerance. The outside tolerance is shown in green on the screen.

Console adjustment

The brightness can be adjusted for both the main console and the remote console. The form is also used to set the display saver's idle time-out.

The MAIN BRIGHTNESS field applies to the main console. The REMOTE SETTING applies to the remote console.

The display saver setting is the amount of time the system is idle before the display is turned off. The idle time may be set from 30 to 120 minutes. The display saver can be disabled during the current power cycle.

Installation Menu

Refer to the Technical Information section for Installation Setup chapter.

Calculator

POSITIP 880's calculator is capable of handling everything from simple arithmetic to complex trigonometry and RPM calculations. The numeric keypad acts as a standard calculator with keys for numbers **0** through **9**, a decimal point, a positive/negative sign (+/-) and **CE** to clear an entry.

The calculator is accessible from the **INFO** soft key. Press the **CALCULATOR** soft key to access the **STANDARD/TRIG** and **RPM** softkeys. Calculations can be entered directly into a highlighted field. See Fig. I.67 & Fig. I.68.

Fig. I.67   Calculator soft keys

Taper calculator functions are available for lathe operations only.

The calculator functions are:

■ Standard/Trigonometry
■ RPM
■ Reference tables
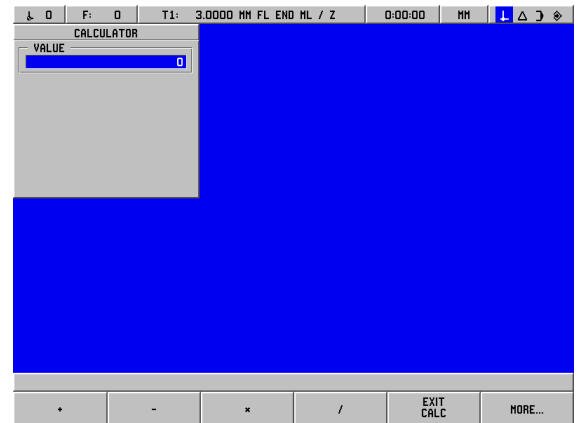
Fig. I.68   1st row calculator soft keys

Math functions (+, –, x, /)

Math functions may be performed separately in the stand-alone calculator or directly in the field you're working in. For example, let's say we need to enter the radius of a circle pocket, but your print only shows the diameter, 6.25. Here's an easy way to figure out the radius:

▶ Highlight the RADIUS field for the circle pattern.

▶ Using the keypad, enter the value for the diameter, 6.25.

▶ Press **INFO.**

▶ Press **CALCULATOR.**

▶ Press the **TRIG** soft key.

▶ Press the **/** soft key.

▶ Press the **2** key.

▶ Press the **ENT** key. The answer, 3.125, will appear in the VALUE field.

▶ Press **USE**. Value will be shown in the RADIUS field.

> ☞ When you need to enter more than one calculation into a numeric field, the calculator will perform multiplication and division before it performs addition and subtraction. If you were to enter 3 + 1 / 8, POSITIP 880 will divide one by eight, then add three for an answer of 3.125.

Trig functions

As we showed above, when you're entering data for a milling or positioning function, your print may not provide you with all of the information you need. If an unknown value can be expressed as a trigonometry function, square root or the square of a number, POSITIP 880 can calculate the dimension for you in the trigonometry calculator. See Fig. I.69 & Fig. I.70.

Using the trig calculator:

▶ While in a numeric field (such as the "TO" field for a line), press the **INFO** soft key, the **CALCULATOR** soft key, and then press the **STANDARD/TRIG** soft key. Press the **MORE** soft key and the soft key row in the picture to the right will appear:

▶ Press the **MORE** soft key to toggle between the three soft key rows.

▶ Enter a number and then press the soft key for the appropriate trigonometry math function.

For example, to enter a radius that has a value equal to the square root of 2, follow this procedure:

▶ Highlight the RADIUS field.

▶ Press the **2** key.

▶ Press **INFO.**

▶ Press **CALCULATOR.**

▶ Press the **TRIG** soft key.

▶ Press the **SQR ROOT** soft key. A value of 1.4142 will appear.
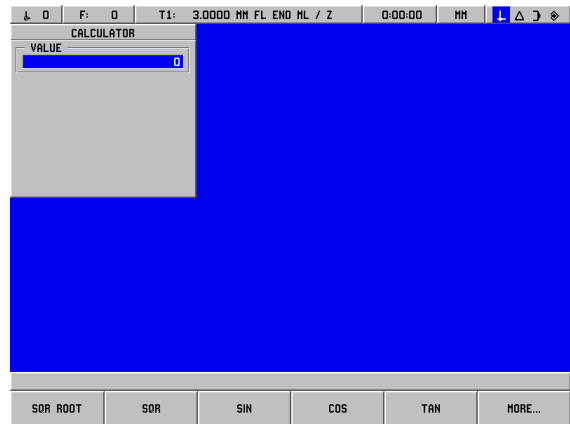
Fig. I.69   2nd row of calculator soft keys
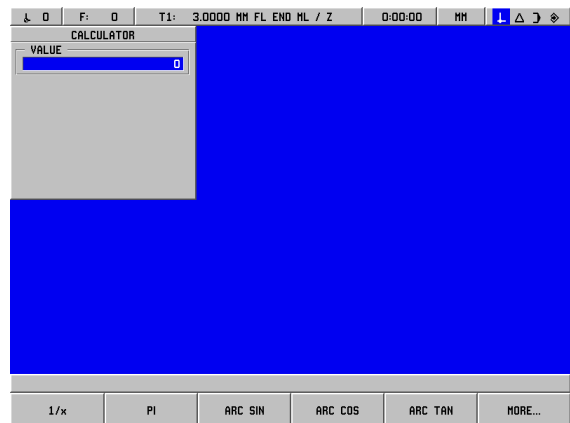
Fig. I.70   Trig function soft keys

Whenever you're calculating the SIN, COS or TAN of an angle, enter the angle first and then press the appropriate soft key. If you were to enter a value of 30 into a numeric field, then press the **SIN** soft key, a value of 0.5 (the SIN of 30 degrees) would appear.

Continuing, let's assume that we are trying to set up a 10 cm SIN bar at a 30° angle and need to know how tall the block underneath it should be. If we multiply 0.5000 by 10, we know that we need to use a 5 cm block.

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

RPM calculator

The RPM calculator is used to set the RPM (or surface cutting speed) based on a specified tool diameter. See Fig. I.71.

▶ 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 diameter if no new value is entered. If there is no current tool, 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.

Surface speed table

▶ 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 either the **FEET/MIN** or **METERS/MIN** soft key depending on the current units.

▶ The RPM CALCULATOR form is closed by pressing the **USE** soft key. There is no return value.

Reference tables

▶ Press the **REFERENCE TABLES** soft key to open on-line help reference tables.
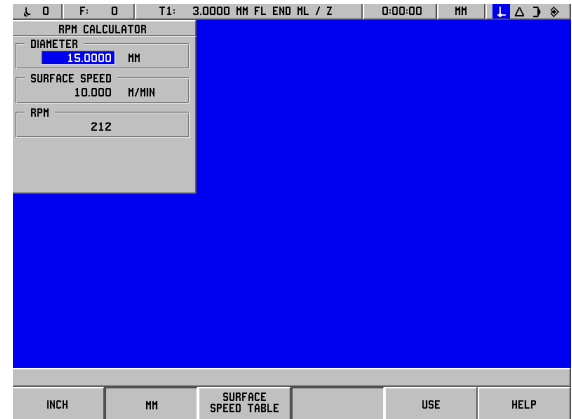
Fig. I.71   RPM CALCULATOR form

Language

The POSITIP 880 supports multiple languages. To change the language selection:

▶ Press **INFO.**

▶ Press the **LANGUAGE** soft key.

▶ Press the **LANGUAGE** soft key until the desired language selection appears on the soft key.

▶ Press **USE** and confirm your selection. The system will restart.

☞ Changing from one language to another will restart the system. This is equivalent to cycling power. It will be necessary to cross over the reference marks to re-establish the current datum settings and reload the current program.

Inch/MM

Positions can be displayed and entered in millimeters or inches. If you choose inches, INCH will be displayed on the status bar at the top of the screen.

To **change** the unit of measure:

▶ Press **INFO.**

▶ Press the **INCH/MM** soft key.

▶ The unit of measure can also be set in JOB SETUP. Refer to Job Setup, chapter I-8.

I – 9 POSITIP 880 Turning Functions

To enter POSITIP 880 for turning functions:

Power Up

Switch on the power (located on the back). It will take approximately 25 - 30 seconds for system to start after power up. The initial screen will appear **(This screen will only appear the first time you power up)**. Select the language by pressing the **LANGUAGE** soft key. See Fig. I.72.

Select the **TURN** soft key to continue with turning functions. You can change the application later in INSTALLATION SETUP under COUNTER SETTINGS.

If you are a novice user to the POSITIP 880 you should review the first 4 sections of this manual for information common to both milling and turning applications.

Fig. I.72   Initial screen

Your POSITIP 880 is now ready for operation and is in the operating mode **ACTUAL VALUE**.
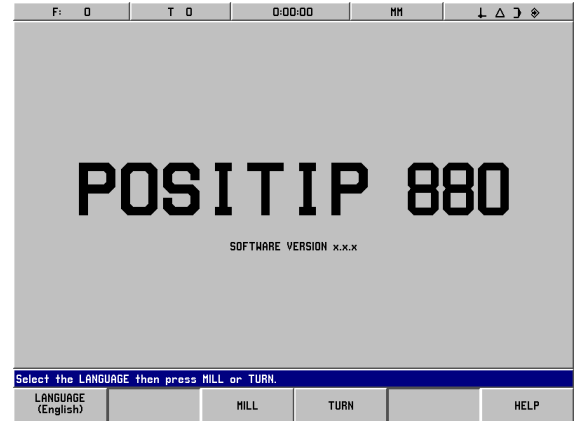
Fundamentals of Positioning

Introduction

The geometry of a workpiece is described by a rectangular or *Cartesian* coordinate system (named in honor of the French mathematician and philosopher René Descartes, in Latin Renatus Cartesius, 1596 to 1650). 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. See Fig. I.73.
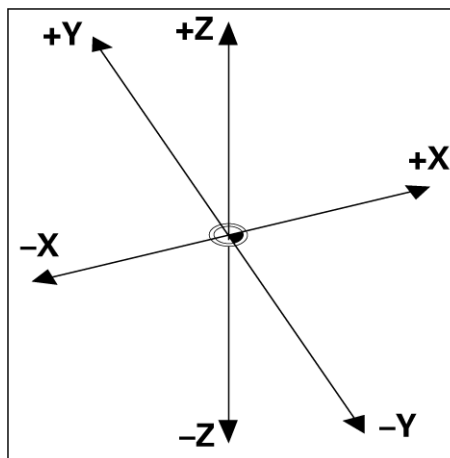
Fig. I.73   Cartesian coordinate system

To determine positions on a workpiece, the coordinate system is "laid" onto the workpiece. With lathe work (i.e. rotationally symmetrical workpieces), the Z axis moves along the axis of rotation while the X axis moves in the direction of the radius or diameter. The Y axis can be disregarded since it would always have the same values as the X axis. See Fig. I.74.
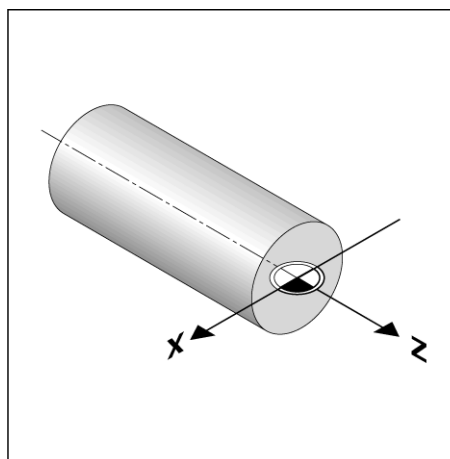
Fig. I.74   The Cartesian coordinate with lathe work

Job Setup for Turning Applications

Two of the features in this JOB SETUP menu are unique to turning applications (DIAMETER AXES, OVERSIZE). See chapter I-8 Job Setup Menu for descriptions of the other general Job Setup features. See Fig. I.75.

The JOB SETUP menu is accessed by pressing the **INFO** soft key, then the **JOB SETUP** soft key. When in the JOB SETUP menu, the following soft keys will be available:

- **INSTALLATION SETUP**
  Press to begin accessing the INSTALLATION SETUP parameters.
- **IMPORT/EXPORT** Press to begin importing or exporting the configuration file.
- **CANCEL CHANGES** Will cancel all changes and exit from JOB SETUP back to the previous mode.
- **USE NEW SETTINGS** Will save any changes to the configuration file before exiting.
- **HELP**
  Will open on-line help.

To access the JOB SETUP menu:

| INFO | Press the **INFO** soft key. |

| JOB SETUP | Press the **JOB SETUP** soft key. |
| | The cursor will default to the first selection on the menu: TOOL TABLE |

Tool table (turning only)

The tool table will support up to 99 tools containing information on each axis. If you have measured your tools using a tool presetter, the offsets may be entered directly. See Fig. I.76.
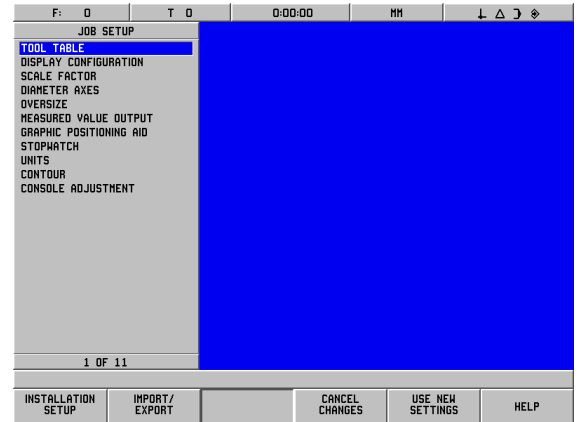
Fig. I.75   JOB SETUP screen in turning

Fig. I.76   Tool table in turning

Tool Table Usage

Example: Entering offsets into the tool table

INFO	Press the **INFO** soft key, then the **JOB SETUP** soft key.
JOB SETUP	

The cursor will default to the TOOL TABLE field.

ENT	Press **ENT**.

TOOL TABLE

↓	ENT	Cursor to the tool you want to define or enter the tool number. Press **ENT**.

I-9 POSITIP 880 Turning Functions

TOOL DESCRIPTION

2 **5**

ENT

If a tool presetter is used to measure the length of a tool, enter the tool offset (25 mm) and press **ENT**.

USE

USE

USE NEW
SETTINGS

Press **USE** soft key twice, then the **USE NEW SETTINGS** soft key once.

☝ The tool table may be protected to prevent changes from being made to the tool information. Refer to II-2 Installation Setup.

Diameter axes

Select DIAMETER AXES to set the axis values to either diameter or radius values depending on the requirements of the drawing. See Fig. I.77.

▶ Cursor to DIAMETER AXES and press **ENT**.

▶ The cursor will be in the X axis field. (The field will default to RADIUS). Depending on the parameter you need for that axis either press the **DIAMETER** soft key or press **ENT** to leave as RADIUS and advance to the next Z AXIS field.

▶ Press **USE**, then **USE NEW SETTINGS**.

Fig. I.77   DIAMETER AXES form

Radius/Diameter Display

Drawings for lathe parts usually give diameter values. POSITIP 880 can display either the radius or the diameter for you. When the diameter is being displayed, the diameter symbol (Ø) is shown next to the position value. See Fig. I.78.

Example: Radius display, position ① X = 20 mm

Diameter display, position ① X = 40$^{\varnothing}$ mm
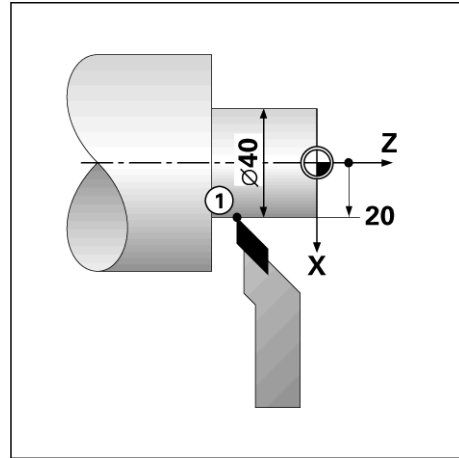
Fig. I.78   Workpiece for radius/diameter display

Oversizes

Oversizes are entered in the JOB SETUP menu. Oversizes are automatically taken into account in the DISTANCE-TO-GO mode. When the displayed distance-to-go is 0, only the oversize amount remains. When you have set the OVERSIZE field to ON, a symbol for oversize appears on the right side of the axis bar. See Fig. I.79.

Oversize symbol: $\bigvee$

Entry sizes for oversize and undersize

Oversize: Positive entry value (up to 999.999 mm).

Undersize: Negative entry value (down to -999.99mm).

Entering oversizes

▶ Cursor to OVERSIZE and press **ENT**.

▶ To enter value either press **ON** soft key or simply begin entering the desired value for that axis. Press **ENT** to advance to the next axis field.

▶ To work without oversizes, either press the **OFF** soft key for each axis field or enter 0 for each field.

▶ Press **USE**, then **USE NEW SETTINGS**.
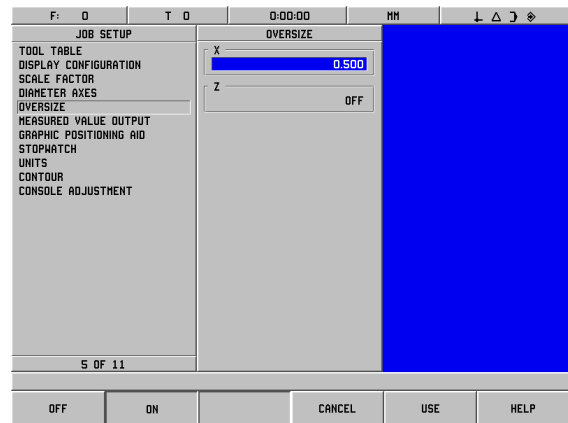
Fig. I.79   OVERSIZE form

Tool Offsetting

The POSITIP 880 can store the dimensional offsets for up to 99 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 or they can be determined using a tool presetter. Refer to the section on Tool Table for more information on entering tool offsets.

The TOOL/SET and NOTE/SET soft keys are available when in the ACTUAL VALUE operating mode. The following outlines the procedures for setting tool offsets using TOOL/SET and NOTE/SET.

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.

Example: Entering tool data when the workpiece diameter is known

Touch the known diameter in the X axis.

Select the axis (X) key or press the **TOOL SET** soft key.

Enter the position of the tool tip, for example, X=10mm. Press **ENT**.

Touch the workpiece face with the tool.

Cursor to the Z axis.

Set the position display for the tool tip to zero, Z=0. Press **ENT**, then press **USE**.

POSITIP 880 stores the tool data under the tool number in the tool table. Set the tool data for all other tools as described here.

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.80.

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:

NOTE/ SET	Press the **NOTE/SET** soft key.
⬤	Select an axis key, such as X.
	Turn the first diameter in the X axis.
NOTE	Press the **NOTE** soft key.
	Retract from the current position.
	Measure the workpiece distance.
1 **5**	Enter the measured distance or radius, for example, 15 mm and press **ENT**.
USE	Press **USE**.
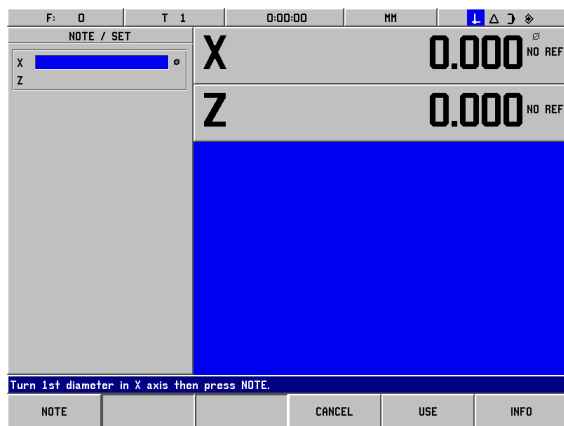
Fig. I.80 NOTE/SET Form

Datum Setting

Example: Setting the workpiece datum (zero point)

The datum is set to zero for the display of the Z axis. All tool data entered are automatically referenced to this datum.

Operating mode: **ACTUAL VALUE**

	Machine the workpiece face. Leave the cutting edge of the tool at the face.
DATUM	Press the **DATUM** soft key.
⬇	Cursor to the Z axis.
0	Enter a value, for example 0. Press **ENT**, then press **USE**.

Taper Calculator

Use the taper calculator to calculate the taper angle. See Fig. I.81.

Entry values:

For the taper ratio, calculation requires:

- Radius of the taper
- Length of the taper

From both diameters and the length, calculations of the:

- Starting diameter
- End diameter
- Length of the taper

Fig. I.81 TAPER CALCULATOR form

INFO	Press the **INFO** soft key.
CALCULATOR	Press the **CALCULATOR** soft key.
TAPER CALCULATOR	You will notice the soft key selection has changed and now includes the TAPER CALCULATOR function. Press the **TAPER CALCULATOR** soft key.

D1/D2 LENGTH

D1/D2/ LENGTH	The soft key in this view toggles between D1/D2/ LENGTH and TAPER RADIUS methods. To calculate the angle using two diameter lengths, press the **D1/ D2 LENGTH** soft key.

Using the numeric keys, enter data into DIAMETER 1 and DIAMETER 2 fields. Press **ENT** after each selection.

Enter data into the LENGTH field, and the angle, based on the calculations entered, will appear in the ANGLE field. |

TAPER RATIO

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

Using the numeric keys, enter data into the RADIUS and LENGTH fields. Press **ENT** after each selection.

The calculated ratio and the angle will appear in their respective fields. |

Programming Turning Functions for POSITIP 880

Program capabilities

Operating mode: **PROGRAMMING**

This section describes the programming capabilities of the POSITIP 880. The operator can load, save or delete a program from internal storage in FILE OPERATIONS. The CLEAR function clears the current program in memory.

Function	Soft key
Allows access to all programming features.	PROGRAMMING FEATURES...
Allows the operator to load, save, delete or clear a program.	FILE OPERATIONS...
Allows the operator access to all block functions.	BLOCK FUNCTIONS...

Fig. I.82   Program editing screen

Programming Features Soft Keys

The PROGRAMMING mode has a **PROGRAMMING** FEATURES soft key which shows all the blocks that can go into a program. Below is a list of **PROGRAMMING FEATURES** soft keys.

Operating mode: **PROGRAMMING**

Function	Soft key
Opens the TOOL CALL block which is used to select a specific tool number from the tool table at run-time.	TOOL CALL
PRESET is used to locate a target position on an axis. The PRESET form allows for entering one or more axis presets. However, multiple axis presets are entered into the program as separate preset blocks.	PRESET
Press to open the MULTIPASS function to turn a shoulder in any number of passes.	MULTI PASS
Opens the LABEL NUMBER form to mark the beginning and the end of a subprogram	LABEL NUMBER
LABEL CALL (REPEAT) is used to call a subprogram 1 or more times. It allows the subprogram to be offset a specified amount on each iteration.	LABEL CALL (REPEAT)
LABEL CALL (MIRROR) allows blocks within a subprogram to be inverted over an axis to represent a mirror image of that subprogram.	LABEL CALL (MIRROR)

Function	Soft key
LINE soft key opens the program form for machining a line.	LINE
ARC soft key opens the program form for machining an arc.	ARC
BLEND soft key opens the program form and NORMAL ARC, INVERTED ARC and CLOSED CONTOUR soft keys for performing a blend.	BLEND
CHAMFER soft key opens the program form and the CLOSED CONTOUR soft key to create a chamfer.	CHAMFER
MORE soft key allows the operator to toggle through the different soft keys rows.	MORE...

Multipass

The multipass cycle enables you to turn a shoulder in any number of passes. See Fig. I.83.

Three blocks are needed to enter into a program:

■ Cycle block
■ X coordinate
■ Z coordinate

The multipass cycle contains all information required for the operation.

Do not delete any blocks from the cycle.

When the program is operating, POSITIP 880 will display the distance-to-go to the two nominal positions immediately following the cycle block. See Fig. I.84.

Example: Turning a shoulder in any number of passes

Workpiece diameter after machining: X = 10 mm

End of shoulder: Z = –30 mm
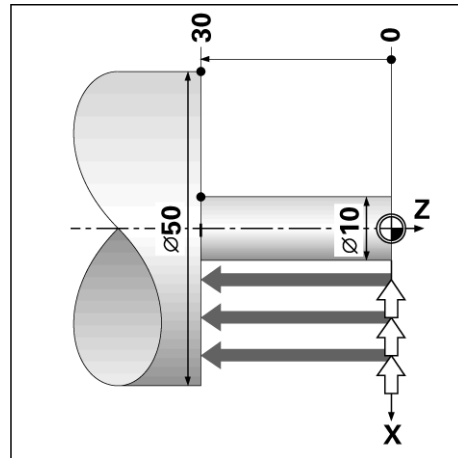
Fig. I.83   Example of MULTIPASS

Operating mode: **PROGRAMMING**

PROGRAMMING FEATURES...	Press the **PROGRAMMING FEATURES** soft key.

MULTI PASS	Press the **MULTIPASS** soft key. The program block MULTIPASS appears on the screen.

X AXIS

1 **0** Enter the nominal position the X axis.

X = 10

Press **ENT** or cursor to the Z axis.

Z AXIS

– **3** **0** Enter the nominal position on the Z axis.

Z= -30

Press **ENT**.

Press **USE**.
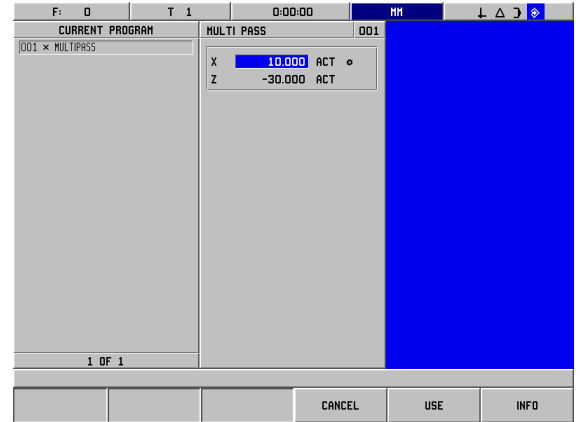
Fig. I.84 Multipass cycle block

File Operation Soft Keys

Function	Soft key
Load from internal storage or to current program memory	LOAD
Save current program in memory to internal storage	SAVE
Delete program from internal storage	DELETE
Clear current program in memory	CLEAR
Select a directory for internal storage	SELECT DIRECTORY
Create a name for a new directory	CREATE DIRECTORY
Delete a directory from internal storage	DELETE DIRECTORY

Function	Soft key
Allows the operator to transfer a program from an external device into POSITIP 880's program memory.	IMPORT PROGRAM
Allows the operator to transfer a program from POSITIP 880 to an external device (PC or printer).	EXPORT PROGRAM

Refer to File Operations in chapter I-6, „Programming POSITIP 880".

Block Function Soft Keys

A block is a single step or a series of steps that make up a program.

Block function soft keys

Operating mode: **PROGRAMMING**

Press the **BLOCK FUNCTIONS** soft key.

Function	Soft key
Change features in a range of blocks	CHANGE BLOCKS
Delete a range of blocks from current program	DELETE BLOCKS
Copy a range of blocks within the current program	COPY BLOCKS
Move a range of blocks within the current program	MOVE BLOCKS
Adjusts the end point of current block and start point of the next block to the intersection point of the two features	NEXT BLOCK
Adjusts the start point of current block and end point of the previous block to the intersection point of the two features	TRIM PREVIOUS
Adjusts the end point of current block and start point of the first block in the path to the intersection point of the two features	TRIM PATH

Refer to Block Functions in chapter I-6, „Programming POSITIP 880".

CHANGING PROGRAM BLOCKS. (SEE FIG. I.85)

OPERATING MODE: **PROGRAMMING**

CHANGE BLOCKS	From within BLOCK FUNCTIONS, press the **CHANGE BLOCKS** soft key.
(ENT)	Select the range of blocks you want to change (FIRST and LAST). Press **ENT** after each selection.
USE	Enter the tool offset, either **OUTSIDE** or **INSIDE**. Press **USE**.
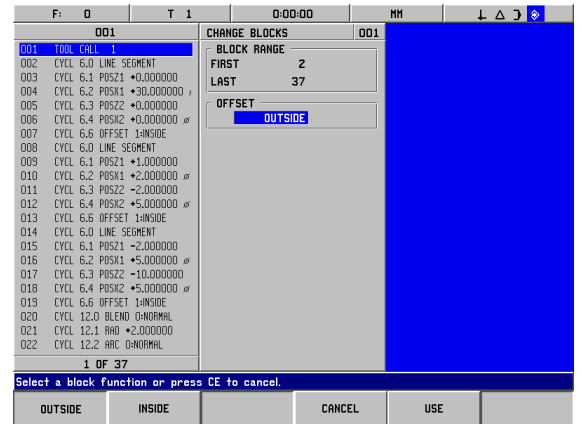
Fig. I.85   Changing a block form

A11:   5.0000

WERKZEUG-TABELLE

4.0000
7.5000
0000
625
75
0

30.1000
35.0050   MM
37.1350   MM
1.1320   MM
1.2600   INCH
22.4000   INCH
INCH

SC
SCH
SCHA
SCHAF
SCHAFT
SCHAFTF

50.0000
45.6500   MM
46.0750   MM
2.6800   MM
21.7000   INCH
1.6200   MM
2810   MM
MM

KUGELFRÄSER
KUGELFRÄSER
KUGELFRÄSER
KUGELFRÄSER

BOHRER
GEWINDEBOHRER
SPITZSENKER

100

II

Technical Information

II – 1 Installation and Electrical Connection

Items Supplied

- POSITIP 880 Display Unit
- Tilt/Swivel Mount
- Power connector
- User's Manual

Mounting Location

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

Installation

M4 screws are used to secure POSITIP 880 from below. For the hole locations: see chapter II-8 Dimensions.

Electrical connection

There are no serviceable items within this unit. Therefore, the POSITIP 880 should not be opened.

The power cord is the primary means of disconnect. 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.

The equipment shall be connected for its intended purpose, or not connected for any use whatsoever.

Electrical requirements

Voltage	100 - 240 Vac
Power	35 VA max.
Frequency	50/60 Hz (+/- 3Hz)
Fuse	800 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 (-22° to 158°F)
Mechanical weight	3.2 kg (7.2 lb.)

Wiring the power connector (See Fig. II.1)

Hot leads: L and N
Ground: ⏚

Minimum diameter of power connection cable: 0.75 mm^2.

Protective earthing

⚠️ It is necessary to connect the protective conductor terminal on the rear panel to the star point of machine ground. Minimum cross-section of the connecting wire: 6 mm^2, see Fig. II.2.

Preventative maintenance

No special preventative maintenance is necessary. For cleaning, wipe lightly with a dry lint-free cloth.
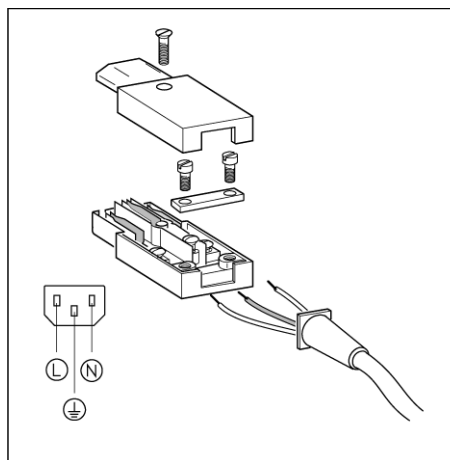
Fig. II.1   Wiring the power connector
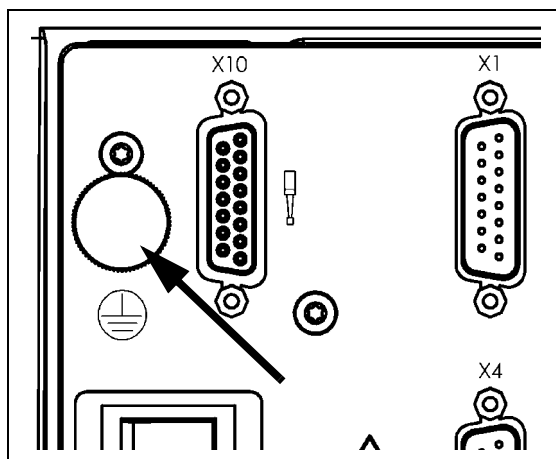
Fig. II.2   The protective conductor terminal on the rear panel.

Connecting the Encoders

POSITIP 880 can be used with **HEIDENHAIN** linear and rotary encoders that provide sinusoidal output (11µApp or 1Vpp) or EnDat (**En**coder **Dat**a). The encoder inputs on the rear panel are designated X1, X2, X3, X4, X5 and X6. See Fig. II.3 & Fig. II.4.

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

Pin layout for POSITIP 880: D-sub connection 15-pin			
D-sub connector 15-pin	Input-signal 11 μA_{PP}	Input-signal 1 V_{PP}	Input-signal EnDat
1	5 V U_P	5 V U_P	5 V U_P
2	0 V U_N	0 V U_N	0 V U_N
3	I_1 +	A +	A +
4	I_1 –	A –	A –
5			DATA
6	I_2 +	B +	B +
7	I_2 –	B –	B –
8			$\overline{DATA}$
9		5 V Sensor	5 V Sensor
10	I_0 +	R +	
11		0 V Sensor	0 V Sensor
12	I_0 –	R –	
13	Internal Shield		Internal Shield
14			CLOCK
15			$\overline{CLOCK}$
Case	External Shield		
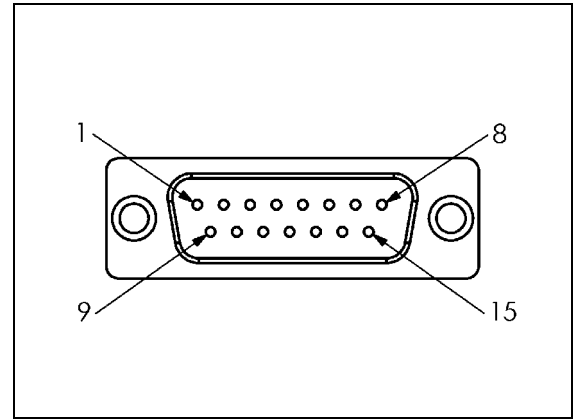
Fig. II.3   15 pin header X1 - X6 plug pinout for encoder signal input.

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

Default configuration:

Encoder input	Mill	Turn
X1	X	X
X2	Y	Z
X3	Z	Xo
X4	W	Zo
X5	U	A
X6	V	Ao
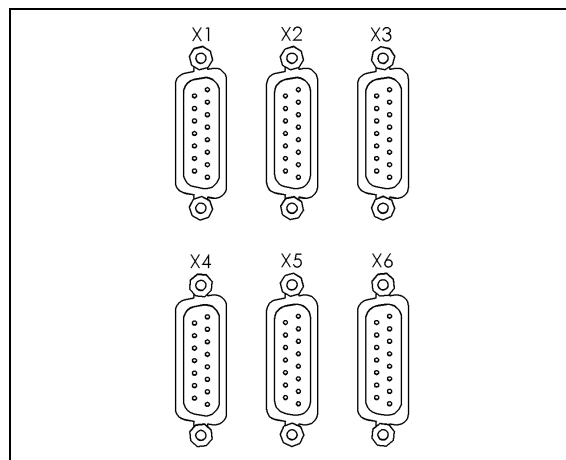
Fig. II.4   Encoder inputs on rear panel.

Connecting an Edge Finder

Connect the **HEIDENHAIN** Edge Finder to the D-sub input X10 on the rear panel.

Adapt POSITIP 880 for use with the Edge Finder through the following operating parameters:

■ Stylus length

■ Stylus diameter

Description of operating parameters see Job Setup chapter.

☞ The operator must enter new edge finder settings.

Pin layout for Edge Finder input (see Fig. II.5 for pinout)

Pin	Assignment	Type
1	Internal shield	
2	Stand-by	KT 130
6	UP +5 V	KT 130
8	UP 0V	KT 130
13	Switch signal	KT 130
Housing	External shield	

All other pins: not assigned
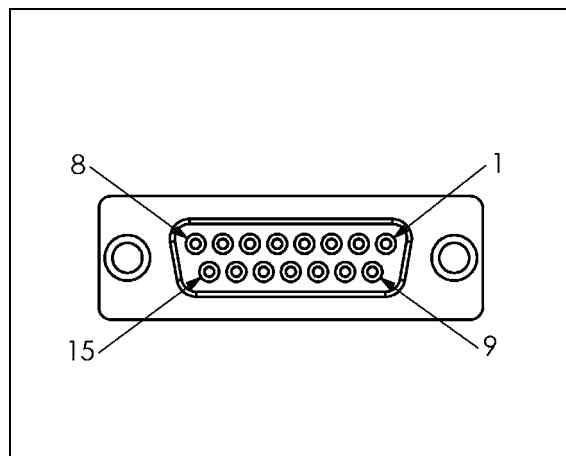
Fig. II.5   15 pin receptacle connector for Edge Finder

II – 2 Installation Setup

Initial switch-on

When you switch on power to your POSITIP 880 for the first time, the screen shown in Fig. II.6 appears. You can now select the appropriate language and the type of application (milling or turning).

Press **LANGUAGE** to scroll through the user interface languages.

For **milling**:

▶ Press the **MILL** soft key.

For **turning**:

▶ Press the **TURN** soft key.

POSITIP 880 automatically provides the functions appropriate to the selected application.

You can change the application later in INSTALLATION SETUP under COUNTER SETTINGS.
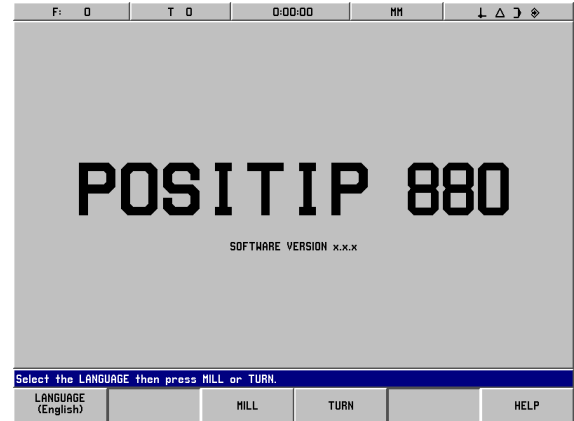
Fig. II.6 POSITIP 880 screen after initial switch-on

This section contains installation setup parameters. Installation setup is accessed by pressing the **INFO** soft key, which brings up **INSTALLATION SETUP** soft key. See Fig. II.7: Installation setup screen

Enter the passcode

Press the **INSTALLATION SETUP** soft key to access the form that requires a passcode to be entered. The **CANCEL** and **USE** soft keys are also available.

Enter the correct passcode (95148) and press the **USE** soft key or **ENT** hard key. (Once you enter the passcode, it will not have to be entered again until you cycle power.) This will open a form containing a list of the INSTALLATION SETUP parameters and the following soft keys:

Fig. II.7 Installation setup screen

Function	Soft key
Press to begin accessing the JOB SETUP parameter.	JOB SETUP
Import or Export configuration file over the serial port.	LOAD
This will cancel all changes and exit from INSTALLATION SETUP back to previous mode.	CANCEL
This will update with new settings and save any changes to the configuration file before it exits.	SAVE
This will open on-screen help.	HELP
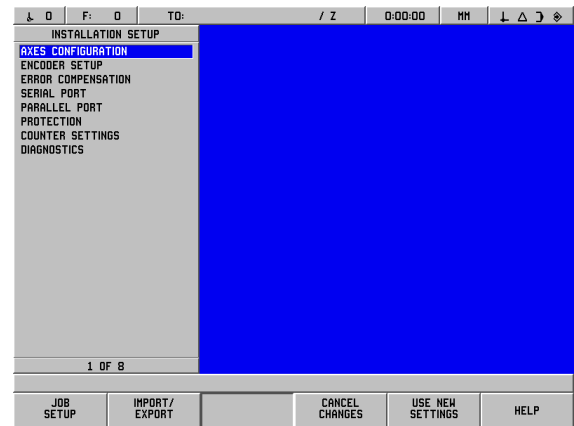
General field/form navigation guide

- Press **Up/Down arrow** keys to move between fields.
- Use keypad to enter numeric values within each field.
- Press **ENT** to confirm the entry within a field.
- Press **USE** when form is completed and save settings to the form.
- Press **USE NEW SETTINGS** to return to the INFO screen.

Axes Configuration

The configuration parameter allows encoder inputs (X1-X6) to be assigned to displayable axes. The axis may consist of a single encoder input or may be a combination of two encoders (2nd encoder position added or subtracted from the 1st). See Fig. II.8 & Fig. II.9.

This is also where the labels for the axes may be changed (A-Z).

To set up the configuration of each axes:

▶ Cursor to AXES CONFIGURATION in the INSTALLATION SETUP menu and press **ENT**.

▶ The AXES CONFIGURATION form is displayed.

▶ Use the numeric keys 1-6 to select the corresponding encoder input for the axis.

▶ Press the **CE** key to remove the encoder input and OFF will be displayed. Which ever encoders are not being used should be switched to the OFF position at this point.

▶ Two encoder inputs can be coupled to an axis by using the numeric keys and the "**+**" and "**-**" soft keys.

▶ Press the **SELECT LABEL** soft key to bring up the SELECT LABEL character chooser form and the **LEFT**, **RIGHT**, **SELECT CHARACTER**, **CANCEL** and **USE** soft keys are available.

▶ Use the **LEFT** and **RIGHT** soft keys and the **Up and Down** arrow keys to highlight the desired character.

▶ Press the **SELECT CHARACTER** soft key or **ENT** hard key to select the character as the axis label and remove the SELECT LABEL form. The soft keys will return to the previous soft key menu.

▶ When you have finished entering values, press the **USE** soft key or the **ENT** hard key to save the changes for that parameter and return to the INSTALLATION SETUP menu.

▶ Graph axes functions are for turning functions only. The default setting for graph axes is to display the Z axis coordinates along the horizontal axis of the part graphic and the X axis coordinates along the vertical axis. If your configuration differs from this, press the **HORIZONTAL** or **VERTICAL** soft keys to select the appropriate axes.
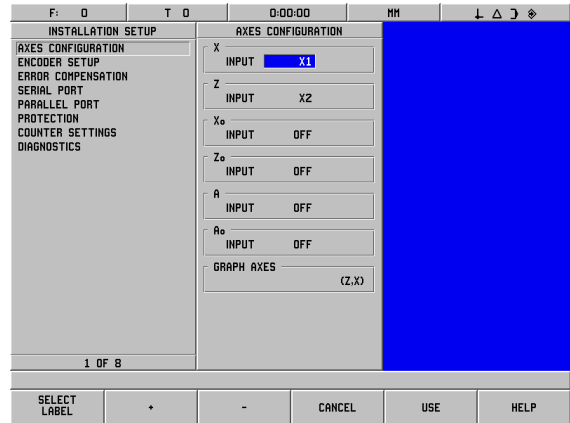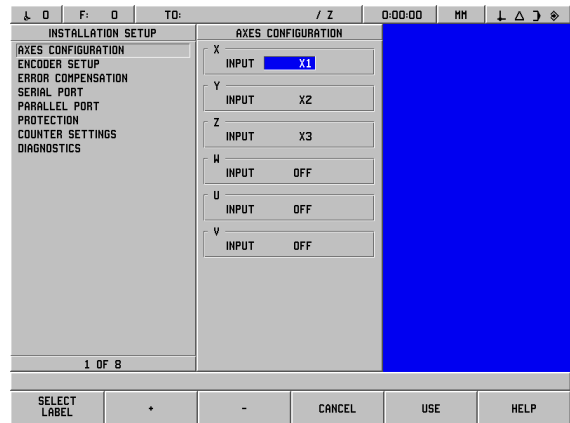

Fig. II.8   AXES CONFIGURATION form (turning)

Fig. II.9   AXES CONFIGURATION form (milling)

Encoder Setup

The ENCODER SETUP is used to set the encoder resolution and type (linear, angular), count direction, reference mark type. See Fig. II.10.

▶ Cursor to the **ENCODER SETUP** to open a list of 6 possible encoders.

▶ Cursor to the encoder you want to change and press **ENT**.

▶ The ENCODER SIGNAL field will already be determined.

▶ Cursor will be in the SIGNAL PERIOD field, select the encoder type by pressing the **ENCODER TYPE** soft key. The default linear encoder signal period is 20 µm.

▶ For linear encoders, use **COARSER** or **FINER** soft keys to select the encoder's signal period in µm (2, 4, 10, 20, 40, 100, 200, 10 240, 12 800) or type in the exact signal period desired. For rotary encoders, enter the encoder's line count. The default encoder type is linear. Refer to II-3 for values.

▶ In the RESOLUTION field, select the **COARSER** or **FINER** resolution value. Default linear resolution is 1 µm.

▶ In the REFERENCE MARK field, select whether the encoder has no reference mark with the **NONE** soft key, a single reference mark with the **SINGLE** soft key or a distance-coded reference mark with the **CODED** soft key.

▶ For coded reference marks, press the **CODED** soft key to select 500, 1000, 2000 or 5000. The default value is CODED with a spacing of 1000. Refer to II-3 for values.

▶ In the 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. The default count direction is POSITIVE.

▶ In the ERROR MONITOR field, select whether signal and slew errors should be monitored by selecting **ON** or **OFF**. The default is ON. When an error message occurs, press the **CE** (Clear Entry) key to remove.

▶ For rotary encoders, an ANGLE DISPLAY field will be available. Select how the angle will be displayed by choosing between three separate soft keys to toggle through the available options. Choose between **0° TO 360°**, **-180° TO +180°**, or **-INF° TO INF°**. The default is 0° TO 360°.

▶ When complete, select **USE**, then cursor to the next encoder input field if necessary.

Fig. II.10 ENCODER SETUP form

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 POSITIP 880 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.11 & Fig. II.12

To calculate the linear error compensation use this formula:

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

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

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

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

LEC = 500 ppm
(rounded to the nearest whole number)

Fig. II.11 Linear error comp., calculation formula

Direct Entry

▶ If known, the encoder's error information may be entered directly. Press the **LINEAR** soft key.

▶ Enter the compensation factor in parts per million (ppm) and press the **USE** soft key.

Autocalculation

▶ This option allows the operator to use gauge blocks to determine the compensation factor.

▶ Press the **AUTOCALCULATION** soft key. The ERROR INTERVAL form will open.

▶ Move the table to the first edge of the gauge block, and press the **START** soft key. The MEASURED field will reset to 0.

▶ Move to the second edge of the gauge block. As the table is moved, the MEASURED field will show the scale's displacement from the first edge.

▶ Press the **END** soft key.

▶ Enter the known gauge block size in the REFERENCE field. If the block edges were approached from opposite directions, include the diameter of the tool or feeler gauge stylus in the REFERENCE VALUE field.

▶ After entering the value, press the **USE** soft key to close the form. The system will calculate the compensation factor.

Fig. II.12 Linear error compensation form

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. POSITIP 880 supports up to 128 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.

▶ Select **NON-LINEAR** and press **EDIT TABLE**.

▶ The encoder's error compensation table and a graph showing the error are displayed in the left two-thirds of the display screen.

▶ The error compensation table shows the table entry, the scale's measured value, and the amount of error at the measured value.

Reading the graph

■ The graph shows a plot of a translation error (reference - measured) 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 blue vertical line. See Fig. II.13.

Starting a Non-linear Error Compensation Table

▶ To start a new error compensation table, first press the **CLEAR TABLE** soft key. This key will only be present if a table containing data already exists.

▶ The error compensation measured values are relative to the table datum location. When the error compensation table is cleared, the POSITIP 880 resets the table datum to the location, of the scale's reference point.

▶ The table datum may be set to another location. Move to the location from which the table values to be measured and press the **SET TABLE DATUM** soft key. The table datum must be set before adding entries to the table.

Configuring the Compensation Table

▶ Error compensation information is added to the table by pressing the **ADD NEW POINT** soft key.

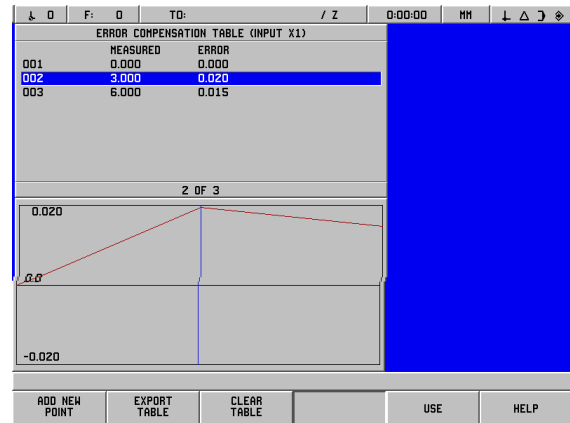▶ There are two possible ways to enter information to the table; Direct Entry or Auto Calculation.

Fig. II.13 Non-linear error compensation form

Direct Entry

▶ If known, the encoder's error information may be entered directly. Press the **DIRECT ENTRY** soft key. An ERROR INTERVAL form will open beside the compensation table.

▶ Enter both the measured value and the error at that value in the form and press the **USE** soft key.

▶ The table and plotted graph will be updated. Pressing the **CANCEL** soft key will close the ERROR INTERVAL form without changing the information in the compensation table.

Autocalculation

▶ This option allows the operator to use gauge blocks to determine the compensation factor.

▶ Press the **AUTOCALCULATION** soft key. The ERROR INTERVAL form will open. Move the table to the first edge of the gauge block, and press the **START** soft key. The MEASURED field will reset to 0.

▶ Move to the second edge of the gauge block. As the table is moved, the Measured field will show the scale's displacement from the first edge.

▶ Press the **END** soft key.

▶ Enter the known gauge block size in the REFERENCE field. If the block edges were approached from opposite directions, include the diameter of the tool or feeler gauge stylus in the REFERENCE VALUE field.

▶ After entering the value, press the **USE** soft key to close the form. The system will calculate the compensation factor.

Importing/Exporting Tables

Error compensation table information for each axis can be imported or exported over the serial port. The tables may also be exported to a printer over the parallel port.

▶ Press the **IMPORT TABLE** soft key to download a compensation table from a PC. This soft key will only be available if the table is empty. If not empty, press **CLEAR TABLE** soft key first.

▶ Press the **EXPORT TABLE** soft key to upload the current compensation table to a PC. It is only available if the table is not empty.

▶ To exit, error compensation table configuration, press the **USE** soft key. The screen will go back to the ERROR COMPENSATION form.

Serial Port (X31)

A printer or computer may be connected to the serial port. Part programs and configuration files may be sent to a printer or computer. Remote commands, remote key codes, part programs and configuration files may be received from a computer. See Fig. II.14.

▶ The BAUD field can be set to 300, 600, 1 200, 2 400, 9 600, 19 200 38 400 57 600 or 115 200 by using the **LOWER** and **HIGHER** soft keys. The default value is 9600.

▶ DATA BITS field can be set to 7 or 8 using the soft keys provided. The default is 8.

▶ The parity can be set to NONE, EVEN, or ODD using the soft keys provided. The default is NONE.

▶ The STOP BITS field can be set to 1 or 2 using soft keys. The Default is 1.

▶ Output tail is the number of carriage returns that will be sent at the end of the transmission. The output tail is initially 0 and can be set to a positive integer value (0 - 9) by using the numeric hard keys.

The serial port settings will be retained on a power cycle. There is no parameter to enable or disable the serial port. A file will only be sent to the serial port if the external device is ready.

Refer to Data Interface section for cable connection and pin assignments.

Fig. II.14 SERIAL PORT form

Parallel Port (X32)

A printer may be connected to the parallel port. See Fig. II.15.

▶ The output tail is initially **0** and can be set to a positive integer value by using the numeric keys (0-9).

▶ The LF AFTER CR field can be set to **ON** or **OFF** using the soft keys. The default is OFF.

The parallel port settings will be retained on a power cycle. There is no parameter to enable or disable the parallel port. A file will only be sent to the parallel port if the external device is ready. If the external device is not ready an error message will appear.

Fig. II.15 PARALLEL PORT form

Protection

The protection feature lets you prevent changes to part programs, the tool table and datum 0. See Fig. II.16.

Two soft keys are available **ON** and **OFF**. The default is OFF. To protect a feature use the **ON** soft key.

When the program protection is set to **ON**, part programs cannot be changed or saved.

Setting the tool table to **ON** prevents it from being changed.

Setting the **datum 0** to **ON** prevents datum 0 from being changed.

The protection settings will be retained on a power cycle.
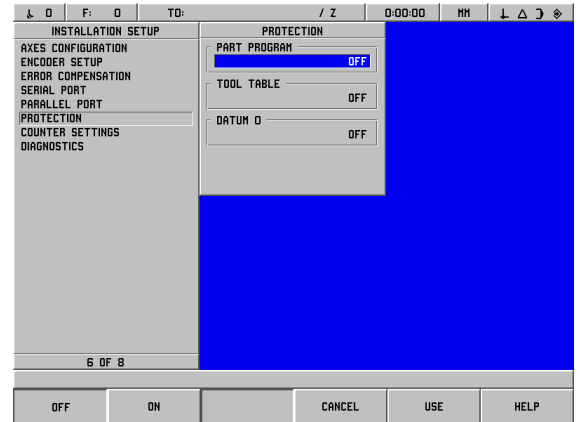
Fig. II.16 PROTECTION form

Counter Settings

The COUNTER SETTINGS feature is the parameter where the operator defines the user application for the readout. The choices are for milling or turning applications. See Fig. II.17.

A **FACTORY DEFAULT** soft key appears in the COUNTER SETTINGS choice of options. 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.

Fig. II.17 COUNTER SETTINGS form

Diagnostics

The DIAGNOSTICS menu provides access for testing keypad, LCD display, encoders and power supply. See Fig. II.18.

Keypad test

An image of the keypad (w/ edge finder) provides 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 **CE** hard key twice to exit the keypad test.

Display test

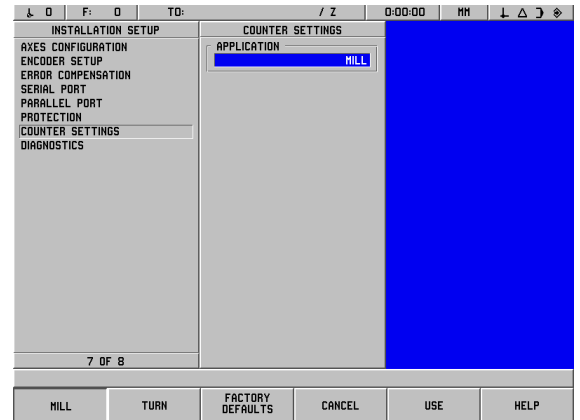Display tests consist of the LCD sleep test and a color test.

▶ Press the **SLEEP TEST** soft key. The screen will go blank, simulating the sleep mode.

▶ To bring the screen back and end the test, press any key.

▶ Press the **COLOR TEST** soft key 6 times to review the complete cycle of colors available.

▶ Press the **EXIT TEST** soft key to leave the display test mode.

Voltage monitor

This allows the operator to monitor the output voltage to encoders. There are no operator controls within this view. This parameter is for the purpose of observation only. When monitoring is completed, press **EXIT TEST**.

Encoder signal plot

This parameter allows the operator to plot the signal of each encoder. See Fig. II.19.

☞ This diagnostic is not available for EnDat encoders.

▶ Select which encoder you want to observe.

▶ Cursor to the desired input and press **ENT**.

▶ As the encoder is moved you will see the signals of the A & B channels.

▶ Press **EXIT TEST** to exit this mode.

System statistics

System statistics displays the amount of time both the display and the system have been on.

It also shows the total distance traveled for each encoder.
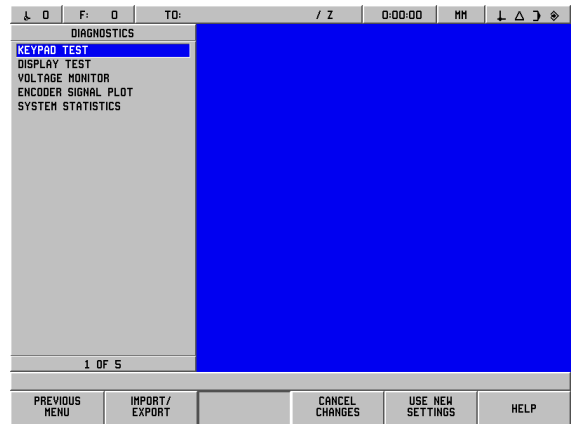
Press **EXIT TEST** to exit the System Statistics mode.
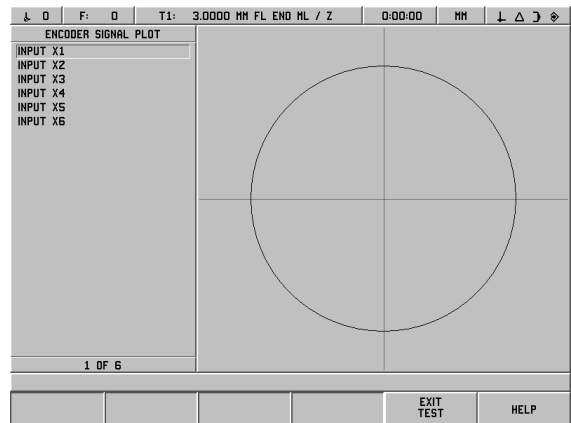

Fig. II.18 DIAGNOSTICS form

Fig. II.19 Example of a signal plot

AMI (Auxiliary Machine Interface)(X51) (Optional)

If AMI hardware is connected to the POSITIP 880, the SWITCHING OUTPUTS form will be available from the installation setup list box. The AMI hardware may be configured with 8 output relays. See Fig. II.20.

To configure an individual relay output, move the cursor to the relay number and press **ENT**. To remove a relay's setting, highlight the relay and press **CE**.

▶ The AXIS field is used to specify which axis position controls the relay. The axis is selected by pressing the **NEXT AXIS** soft key.

The DIRECTIONAL soft key is used to select the directional signal mode. When selected, the relay's operation is based on the position value's sign. The relay is activated when the value is non-negative. The relay is de-activated when the value is negative. If DIRECTIONAL is selected, the axis selection may be set to ALL AXES. In this configuration, the relay's output is set when an operation sets any of the distance-to-go mode display values.

▶ The CONDITION field is used to specify the required relationship between the current position display and the switching point. It is set by pressing the soft keys.

▶ The PRESET field is used to define the switching points and to select whether the point is an actual value or distance-to-go position. The position source is selected by pressing one of the soft keys.

▶ The POINT/RANGE field is used to define whether the condition refers to a point on the axis or refers to range about zero. If a range is specified, the sign of the value is ignored.

▶ To fill in the RELAY ON/OFF field press the **OFF** or **ON** soft keys. When the condition has been satisfied the relay will turn OFF or ON.

▶ The RELAY OPERATION field is used to further define how the relay will operate when the switching condition is met. The options include CONTINUOUS, PULSED or SINGLE. SINGLE is only an option when the distance-to-go position is selected in the PRESET field.

In CONTINUOUS mode, the relay remains activated (ON or OFF) as long as the switching condition is met.

In PULSED mode, the relay is activated only for a predefined, period of time. After the period has timed out, the relay is de-activated. The time is entered in the RELAY OPERATION field.

If the **PULSED** soft key was pressed, a delay value must be entered using the numeric keypad.

In the SINGLE mode, the relay will transition to the activated state only one time after the axis value is set. It only applies to distance-to-go mode position values.

The AMI settings are retained on a power cycle.

Fig. II.20 AMI form

II Technical Information

Remote Console (X61) (Optional)

The remote console functions as duplicate POSITIP 880. The same images display upon both consoles and either keypad may be used for any function. The POSITIP 880's remote console is interfaced via X61. The remote console is equipped with an interface cable that connects to the POSITIP 880's remote console connection (X61).
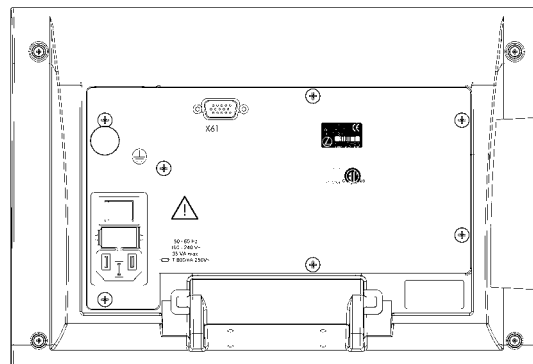
Fig. II.21 Back view of remote console

II – 3 Encoders and Measured Value Display

These tables describe all operating parameters which you must set for the encoders and measured value display. Most entries can be found in the operating instructions for your encoder.

Example settings for HEIDENHAIN linear encoders with 11-μA_{PP} signals

Encoder	Signal period	Reference marks
CT MT xx01	2	Single
LS 303/303C LS 603/603C	20	Single/ 1000
LS 106/106C LS 406/406C	20	Single/1000
LB 302/302C	40	Single/2000
LIM 501	10 240	Single

Example settings for HEIDENHAIN linear encoders with 1-V_{PP} signals

Encoder	Signal period	Reference marks
LIP 382	0.128	–
MT xx81 LIP 481A/481R	2	Single
LIP 481X	2	Single
LF 183/183C LF 481/481C LIF 181/181C LIP 581/581C	4	Single/5000
LS 186/186C	20	Single/1000
LB 382/382C LIDA 18x/18xC	40	Single/ 2000
VM 182	4	–
LIDA 10x/10xC	100	Single/1000
LIM 581	10 240	Single

Setting the display step with rotary encoders

With rotary encoders, the display step depends on the

- line count of the encoder and the
- angle display step

Both parameters are entered separately for each rotary axis.

Example settings for HEIDENHAIN rotary encoders

Encoder	Line count	Reference marks
ROD 250, RON 255	9 000/18 000	1
ROD 250C, RON 255C	9 000	500
ROD 250C, ROD 255C ROD 700C, RON 705C RON 706C	18 000	1 000
ROD 700C, ROD 800C	36 000	1 000

II – 4 Data Interface

The POSITIP 880's data interface includes the RS-232 serial port (X31) and parallel port (X32). The serial port supports bi-directional data communications which allows data to be exported to or imported from an external device. The parallel port only allows data to be exported to an external device.

Data which can be exported from the POSITIP 880 to an external serial or parallel device includes:

■ User part programs

■ Job and installation configuration parameters

■ Non-linear error compensation tables

■ Measured value output (display values or probing functions)

Data which can be imported into POSITIP 880 from an external device includes:

■ User part programs

■ Job and installation configuration parameters

■ Non-linear error compensation tables

■ Remote key commands

This chapter covers what you need to know about **setting up** the data interface:

■ Pin layout of data interface

■ Signal level

■ Wiring of the connecting cable and connectors

■ Baud rate (data transfer speed)

■ Data format

Serial port

The RS-232-C/V.24 serial port is located on the rear panel (see Fig. II.22 for pin layout). The following devices can be connected to this port:

∎ Printer with serial data interface
∎ Personal computer with serial data interface

For operations that support data transfer, import and export soft keys will be available. Refer to the operation section of the manual for more information.

To export data to a serial port printer, press the **EXPORT** soft key. The data is exported in an ASCII text format which can be printed directly.

To export or import data between the POSITIP 880 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 POSITIP 880 and the PC is in ASCII text format.

To export data from the POSITIP 880 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 POSITIP 880's **EXPORT** soft key.

To import data into the POSITIP 880 from a PC, the POSITIP 880 must first be made ready to receive the data. Press the POSITIP 880's **IMPORT** soft key. After the POSITIP 880 is ready, setup the terminal communications program on the PC to send the desired file in ASCII text format.
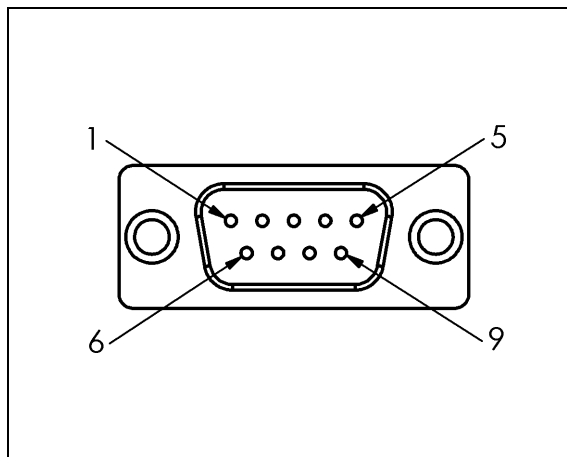
> ☞ The POSITIP 880 does not support communication protocols such as Kermit or Xmodem.

Fig. II.22 Pin layout of RS-232-C/V.24 data interface

Wiring the connecting cable

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

Full wiring

Before the POSITIP 880 and your PC can communicate, they need to be connected to each other with a serial cable. See Fig. II.23 & Fig. II.24.
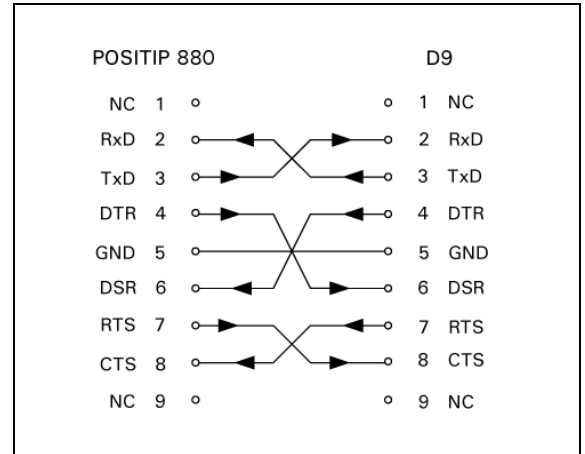
Fig. II.23 Pin connection for serial port with handshaking

Pin assignment

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

Fig. II.24 Pin connection for Serial Port without handshaking

Signal

Signal	Signal level "1"= "active"	Signal level "0"= "inactive"
TXD, RXD	-3 V to -15 V	+ 3 V to + 15 V
RTS, CTS DSR, DTR	+ 3 V to + 15 V	-3 V to -15 V

External Operations via RS-232 Data Interface

You can operate the display unit over the RS-232 data interface using an external device. The following key commands are available:

Format	
<ESC>TXXXX<CR>	Key is pressed
<ESC>AXXXX<CR>	Output of screen contents
<ESC>SXXXX<CR>	Special functions

Sequence of commands	Function
<ESC>T0000<CR>	Key '0'
<ESC>T0001<CR>	Key '1'
<ESC>T0002<CR>	Key '2'
<ESC>T0003<CR>	Key '3'
<ESC>T0004<CR>	Key '4'
<ESC>T0005<CR>	Key '5'
<ESC>T0006<CR>	Key '6'
<ESC>T0007<CR>	Key '7'
<ESC>T0008<CR>	Key '8'
<ESC>T0009<CR>	Key '9'
<ESC>T0100<CR>	Key 'CE' or 'CL'
<ESC>T0101<CR>	Key '-'
<ESC>T0102<CR>	Key '.'
<ESC>T0104<CR>	Key 'ENT'
<ESC>T0109<CR>	Key 'X'
<ESC>T0110<CR>	Key 'Y'/'Z'/'Z_0'
<ESC>T0111<CR>	Key 'Z'
<ESC>T0112<CR>	Key 'IV''
<ESC>T0114<CR>	Key 'Soft key 1'
<ESC>T0115<CR>	Key 'Soft key 2'
<ESC>T0116<CR>	Key 'Soft key 3'
<ESC>T0117<CR>	Key 'Soft key 4'
<ESC>T0118<CR>	Key 'Soft key 5'
<ESC>T0119<CR>	Key 'Soft key 6'
<ESC>T0131<CR>	Key 'Nominal Value'
<ESC>T0132<CR>	Key 'Distance-to-go'
<ESC>T0133<CR>	Key 'PGM-Execution'
<ESC>T0134<CR>	Key 'PGM-Edit
<ESC>T0137<CR>	Key 'Cursor Up'
<ESC>T0138<CR>	Key 'Cursor Down'

Sequence of commands	Function
<ESC>T0140<CR>	Key 'I' (incremental)
<ESC>A0000<CR>	Send device identification
<ESC>A0200<CR>	Send actual position
<ESC>S0000<CR>	Reset device
<ESC>S0001<CR>	Lock keyboard
<ESC>S0002<CR>	Release keyboard

Parallel port

The parallel port is located on the rear panel (see Fig. II.25 & Fig. II.26). The following devices can be connected to this port:

■ Printer with "Centronics" parallel data interface.

■ Printer that can communicate in "Compatability Mode" (also commonly called Centronics Mode), e.g. Okidata 182 Dot Matrix, HP Laser Jet II.

For operations that support data transfer, import and export soft keys will be available. Refer to the operation section of the manual for more information.

To export data to a parallel port printer, press the **EXPORT** soft key. The data is exported in an ASCII text format which can be printed directly.
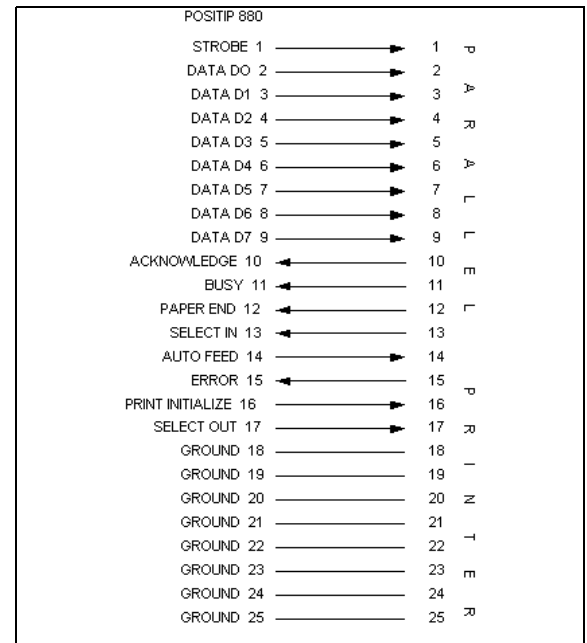

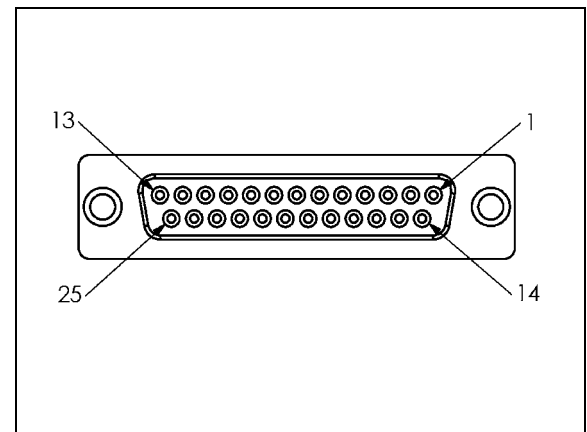Fig. II.25 Pin connection for parallel port

Fig. II.26 25 Pin receptacle X32

II – 5 Measured Value Output

Examples of character output at the data interface

If you have a PC you can retrieve values from POSITIP 880. For activation, see Measured Value Output under Job Setup. In all three examples, measured value output is started with **Ctrl B** (sent over serial interface) or a **switching signal at the EXT input** (within optional Auxiliary Machine Interface). **Ctrl B** will transmit the currently displayed values in either DRO 1 or DRO2, whichever is currently visible.

Data output using external signals

Example 1: Linear axis with radius display X = + 41.29 mm

X	=	+	4 1	.	2 9		R	<CR>	<LF>
1	**2**	**3**	**4**	**5**	**6**	**7**	**8**	**9**	**10**

1 Coordinate axis

2 Equal sign

3 +/– sign

4 2 to 7 places before the decimal point

5 Decimal point

6 1 to 6 places after the decimal point

7 Unit: blank space for mm, **"** for inches

8 Actual value 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)

Data output using Edge Finder

In the next three examples, measured value output is started with a **switching signal from the edge finder.** Printing capability can be turned on or off in the Job Setup parameter Measured Value Output. Information from here is transmitted from the selected axis.

II Technical Information

Example 4: Probing function Edge Y = –3674.4498 mm

Y		:	-	3 6 7 4	.	4 4 9 8		R	<CR>	<LF>
1	**2**	**3**	**4**	**5**	**6**	**7**	**8**	**9**	**10**	**11**

1 Coordinate axis

2 2 blank spaces

3 Colon

4 +/– sign or blank space

5 2 to 7 places before the decimal point

6 Decimal point

7 1 to 6 places after the decimal point

8 Unit: blank space for mm, **"** for inches

9 **R** for radius display, **D** for diameter display

10 Carriage return

11 Blank line (Line Feed)

Example 5: Probing function Centerline

Coordinate of centerline on X axis CLX = + 3476.9963 mm (**C**enter **L**ine **X** axis)

Distance between the probed edges DST = 2853.0012 mm (**Dist**ance)

CLX	:	+	3 4 7 6	.	9 9 6 3		R	<CR>	<LF>
DST	:		2 8 5 3	.	0 0 1 2		R	<CR>	<LF>
	1	**2**	**3**	**4**	**5**	**6**	**7**	**8**	**9**

1 Colon

2 +/– sign or blank space

3 2 to 7 places before the decimal point

4 Decimal point

5 1 to 6 places after the decimal point

6 Unit: blank space for mm, " for inches

7 R for radius display, D for diameter display

8 Carriage return

9 Blank line (Line Feed)

Example 6: Probing function Circle Center

First center point coordinate, e.g. CCX = –1616.3429 mm, Second center point coordinate, e.g. CCY = +4362.9876 mm, (**C**ircle **C**enter **X** axis, **C**ircle **C**enter **Y** axis; coordinates depend on working plane) Circle diameter DIA = 1250.0500 mm

CCX	:	-	1 6 1 6	.	3 4 2 9		R	<CR>	<LF>
CCY	:	+	4 3 6 2	.	9 8 7 6		R	<CR>	<LF>
DIA	:		1 2 5 0	.	0 5 0 0		R	<CR>	<LF>
	1	**2**	**3**	**4**	**5**	**6**	**7**	**8**	**9**

1 Colon

2 +/– sign or blank space

3 2 to 7 places before the decimal point

4 Decimal point

5 1 to 6 places after the decimal point

6 Unit: blank space for mm, " for inches

7 R for radius display, D for diameter display

8 Carriage return

9 Blank line (Line Feed)

II – 6 Specifications for Milling

POSITIP 880 Data	
Axes	Up to 6 axes from A - Z
Encoder inputs	6 x ⌒⌄ 11 µA$_{PP}$, 1 V$_{PP}$ or EnDat (switchable); input frequency max. 100 kHz for incremental or absolute HEIDENHAIN encoders
	▪ ***Signal period:*** 0.128 µm, 2 µm, 4 µm, 10 µm, 20 µm, 40 µm, 100 µm, 10 240 µm
	▪ ***Line count:*** 9 000/18 000/36 000
Subdivision factor	Max. 1 024-fold
Display step	Linear axes: 1 mm to 0.005 µm Rotary axes: 1° to 0.0001° (00°00'01'')
Display	Color flat-panel display for position values, dialog and input display, graphic functions, graphic positioning aid
	▪ ***Multi-lingual text***
	▪ ***Status display:*** Operating mode, REF, inches, scaling factor, feed rate Datum number Tool number and tool axis Tool compensation R–, R+, R0
Functions	▪ REF reference-mark evaluation for distance-coded or single reference marks
	▪ Distance-to-go mode, nominal position input (absolute or incremental)
	▪ Contour monitoring with magnify function
	▪ Scaling factor
	▪ **HELP:** On-screen operating instructions
	▪ **INFO:** Calculator, cutting data calculator, user and operating parameters
	▪ 99 datum points and 99 tools
	▪ Probing functions for datum acquisition, preferably with the KT Edge Finder: Edge, Center Line and Circle Center
	▪ Tool radius compensation
	▪ Calculation of positions for bolt hole circles and linear hole patterns
	▪ Positioning aids for milling and the roughing out of rectangle pockets
Programming	Up to 999 program blocks in each program; sub programming capability; teach-in programming
	▪ ***Cycles:*** Lines, arcs, chamfers, circle patterns, linear patterns and rectangle pockets
Error compensation	Linear and non-linear, up to 128 measuring points

POSITIP 880 Data

Data interface	■ *Serial:* **RS-232-C/V.24** 300 to 115 200 baud For output of programs, measured values and parameters; For input of programs and parameters ■ *Parallel:* **Centronics** for output of measured values
Switching inputs/outputs	Upon request (via separate AMI input/output unit)
Accessories	■ Tilting base ■ Remote console ■ KT 130 Edge Finder
Main power input	100 V to 240 V (–15 % to +10 %) 50 Hz to 60 Hz (±3 Hz); power consumption 35 W
Operating temperature	0 °C to 45 °C (32 °F to 113 °F)
Degree of protection (EN 60529)	IP 40 (IP 54 front panel)
Weight	3.2 kg

II – 7 Specifications for Turning

POSITIP 880 Data	
Axes	Up to 6 axes from A to Z, Z_0, Z_S
Encoder inputs	6 x $\curvearrowleft$ 11 µA$_{PP}$, 1 V$_{PP}$ or EnDat (switchable); input frequency max. 100 kHz for incremental or absolute HEIDENHAIN encoders
	■ *Signal period:* 0.128 µm, 2 µm, 4 µm, 10 µm, 20 µm, 40 µm, 100 µm, 10 240 µm
	■ *Line count:* 9 000/18 000/36 000
Subdivision factor	Max. 1 024-fold
Display step	Linear axes: 1 mm to 0.005 µm Rotary axes: 1° to 0.0001° (00°00′01″)
Display	Color flat-panel display for position values, dialog and input display, graphic functions, graphic positioning aid
	■ *Multi-lingual text*
	■ *Status display:* Tool number Diameter display Ø Sum display
Functions	■ REF reference-mark evaluation for distance-coded or single reference marks
	■ Distance-to-go mode, nominal position input (absolute or incremental)
	■ Contour monitoring with magnify function
	■ Scaling factor
	■ **HELP:** On-screen operating instructions
	■ **INFO:** Calculator, taper calculator, user and operating parameters
	■ 1 datum point, 99 tools
	■ Freezing tool position for back-off
	■ Oversize allowance
Programming	Up to 999 program blocks in each program; sub programming capability; teach-in programming
	■ *Cycles:* Lines, arcs, chamfers, multipass turning, blend
Error Compensation	Linear and non-linear, up to 128 measuring points
Data interface	■ *Serial:* **RS-232-C/V.24** 300 to 115 200 baud For output of programs, measured values and parameters; For input of programs and parameters
	■ *Parallel:* **Centronics** for output of measured values
Switching inputs/outputs	Upon request (via separate AMI input/output unit)

POSITIP 880 Data	
Accessories	■ Tilting base ■ Remote console
Main power input	100 V to 240 V (–15 % to +10 %) 50 Hz to 60 Hz (±3 Hz); power consumption 35 W
Operating temperature	0 °C to 45 °C (32 °F to 113 °F)
Degree of protection (EN 60 529)	IP 40 (IP 54 front panel)
Weight	3.2 kg

II Technical Information

II – 8 Dimensions

287.05
11.30

194.67
7.66

Front view with Dimensions

Back view

SECTION A-A

25.02
.985

M4 THREAD

50.01
1.969

Bottom view with Dimensions

A

A

Side view
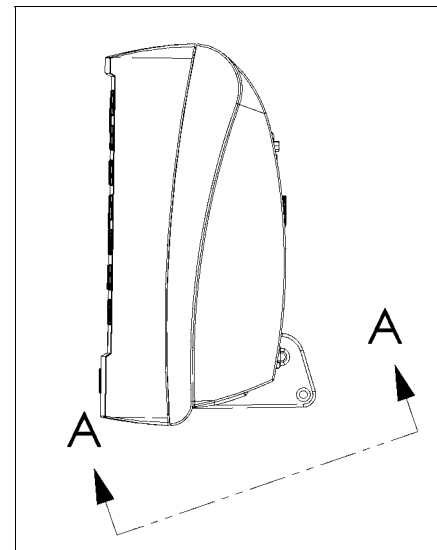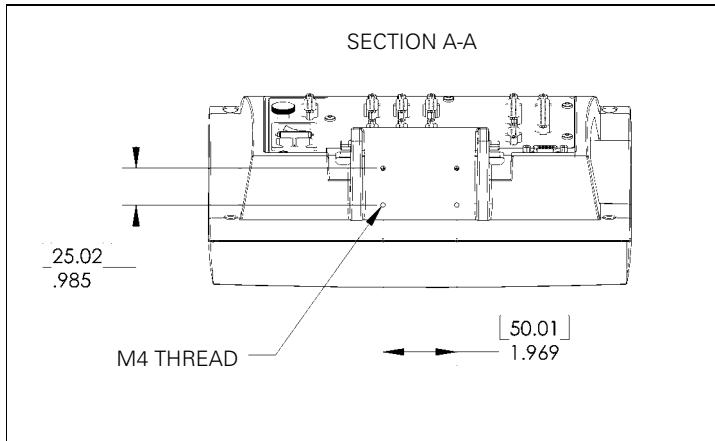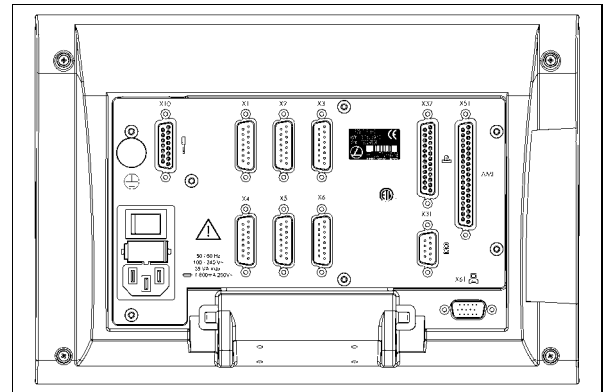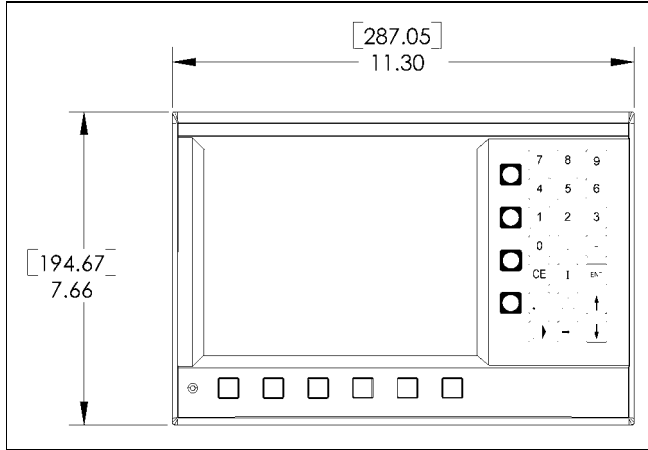
II – 9 Accessories

Accessory ID Numbers

ID Number	Accessories
382891-01	Pkgd, POSITIP 880 Brkt Kit, Tilt Swivel Mnt.
382929-01	Pkgd, Universal Mounting Arm, POSITIP 880
382892-01	Pkgd, Mounting Base POSITIP 880
378039-01	Pkgd, Remote Main Assy POSITIP 880
382930-01	AMI Box
382898-05	Pkgd, Remote cable 5 M
382898-10	Pkgd, Remote cable 10 M
382898-20	Pkgd, Remote cable 20 M
382898-30	Pkgd, Remote cable 30 M

II Technical Information

POSITIP 880 Mounting Instructions
Universal Mounting Arm
Id. Nr. 382 929-01

HHCS 5/8-11 x 3"
SCSS 5/8-2" x 1/2-13 x .8"
HHCS m16 x 75 mm
HHCS m14 x 75 mm

5/8" Flat Washer (2)

Tilt/Swivel Assembly

M6 x 12 mm SHSS (4)

Mounting Base

Arm

M6 x 12 mm SHSS (4)

Yoke

M6 x 12 mm SHSS (4)

M8 x 70 mm HHCS

Flat Washer

M6 x 30 mm SHCS (2)
or
1/4-20 x 1-1/4" SHCS (2)

M8 Locking Hex Nut

End Cap (2)

Tilt/Swivel Assembly
P/N 382891-01

Arm

44.45
1.75

Front View of Yoke

End Cap (2)
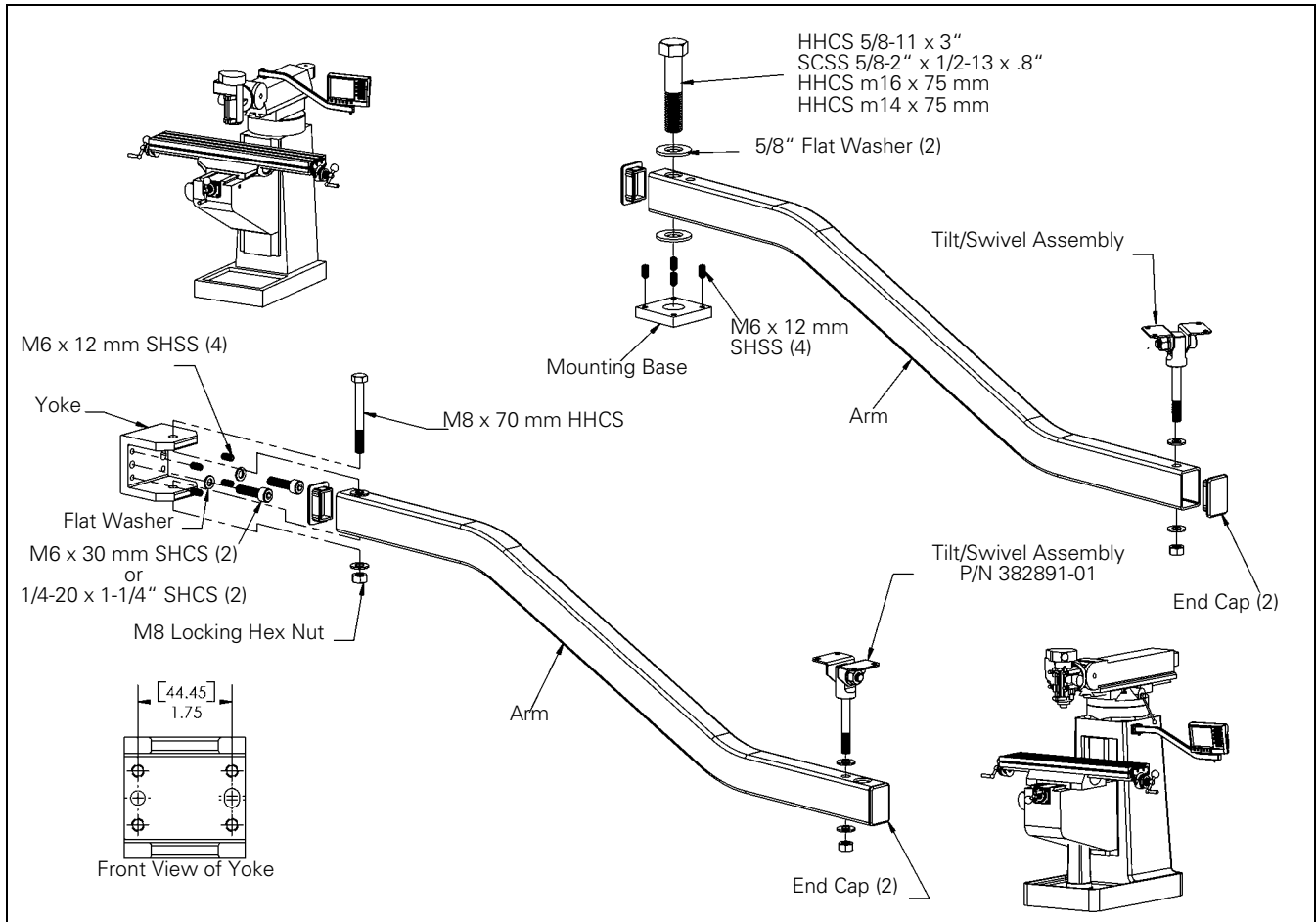
POSITIP 880 Mounting Instructions
Mounting base
Id. Nr. 382 892-01

POSITIP 880

M5 Shoulder Screw

M5 Flat Washer

M5 BHCS

Spacer Washer

Mounting Base

203.20
8.00 TYP

4X M5

158.75
6.25 TYP
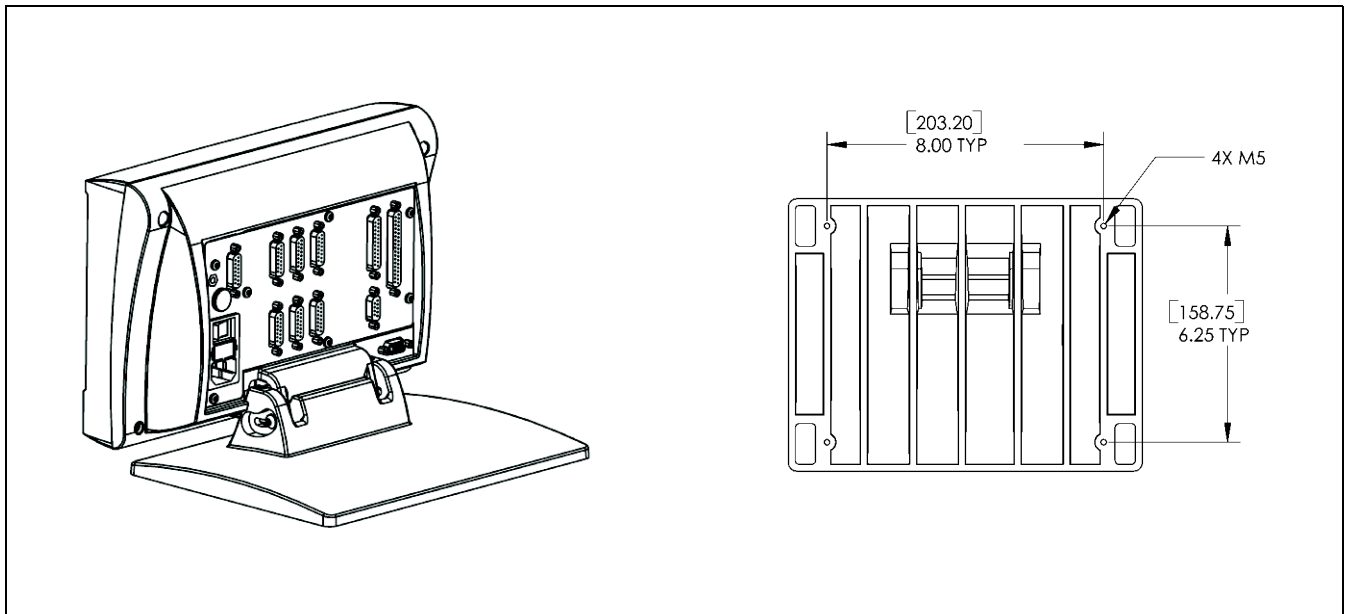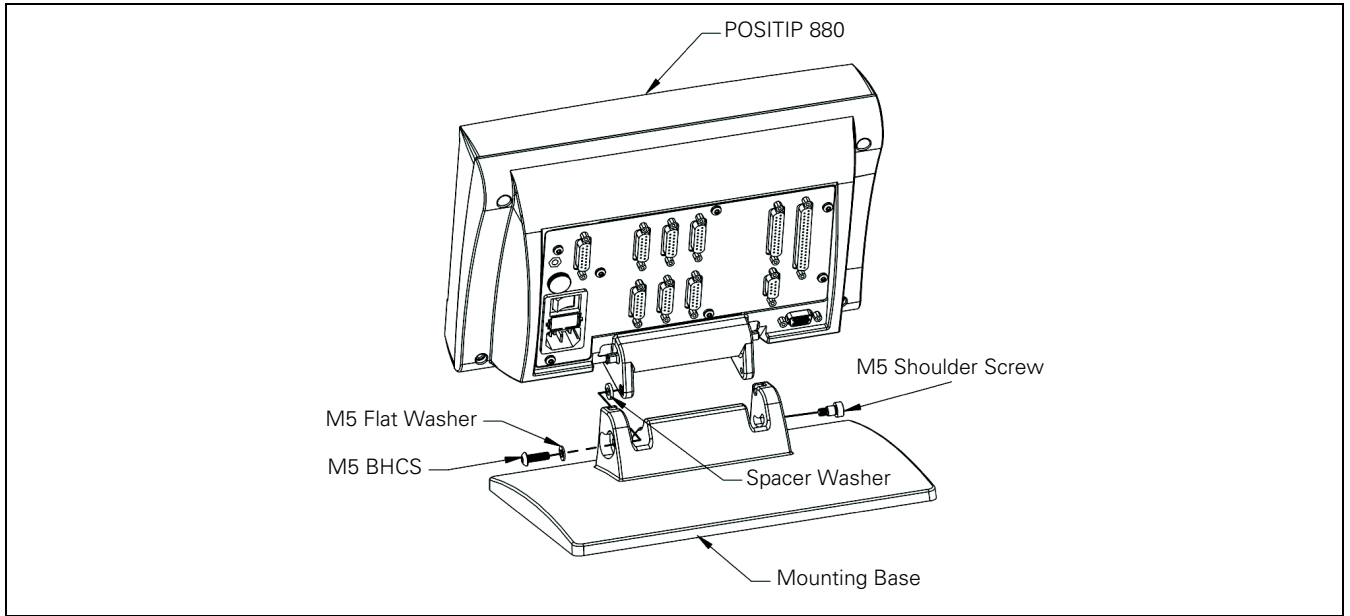
POSITIP 880 Mounting Instructions
Tilt/Swivel Mount
Id. Nr. 382 891-01

M8 x 30 SHCS

Bracket (2)

M8 Flat Washer (4)

M8 x 55 mm Swing Bolt

M8 Locking Hex Nut (2)

4 mm BHCS (4)

Index

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

Technical support FAX +49 (86 69) 31-10 00
    e-mail: service@heidenhain.de
Measuring systems ☎ +49 (86 69) 31-31 04
    e-mail: service.ms-support@heidenhain.de
TNC support ☎ +49 (86 69) 31-31 01
    e-mail: service.nc-support@heidenhain.de
NC programming ☎ +49 (86 69) 31-31 03
    e-mail: service.nc-pgm@heidenhain.de
PLC programming ☎ +49 (86 69) 31-31 02
    e-mail: service.plc@heidenhain.de
Lathe controls ☎ +49 (7 11) 95 28 03-0
    e-mail: service.hsf@heidenhain.de

www.heidenhain.de