HEIDENHAIN

User's Manual

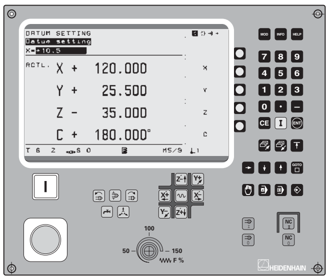
TNC 124

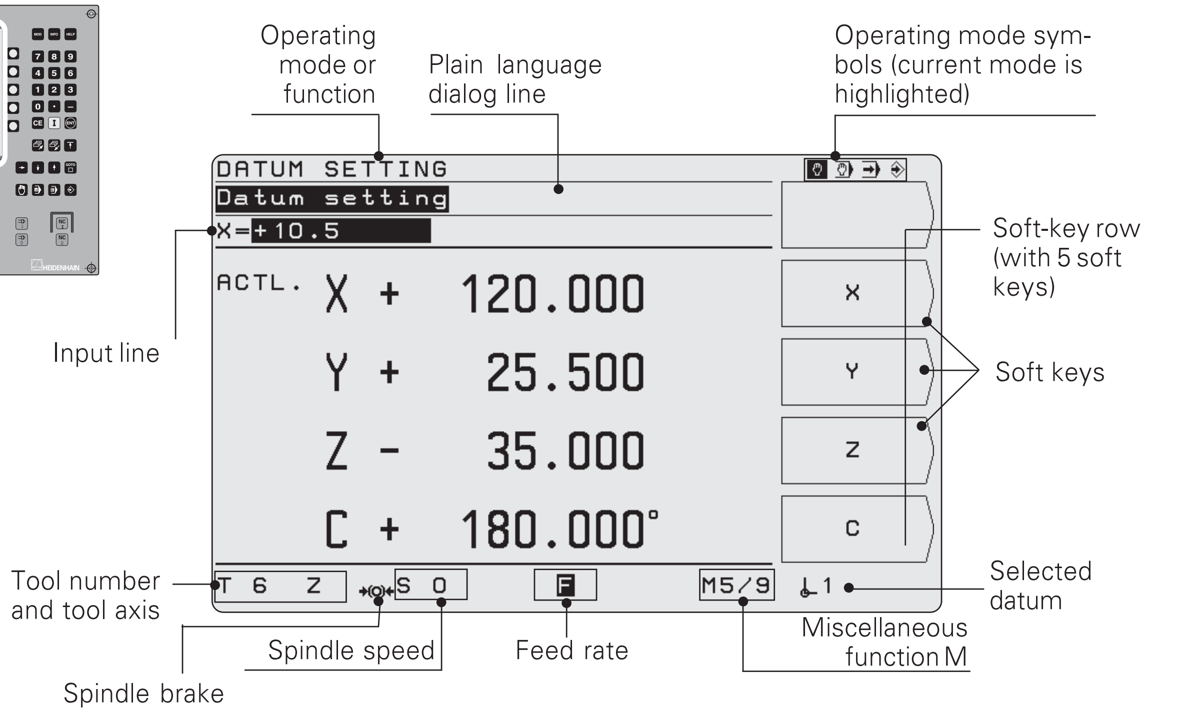
July 2004

TNC Guideline:
From workpiece drawing to program-controlled machining

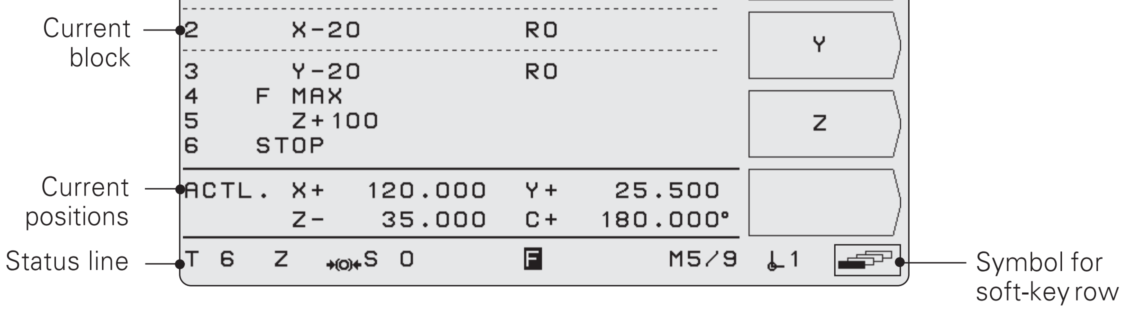
Step	Task	TNC operating mode	Starting on page
	Preparation		
1	Select tools	—	—
2	Set workpiece datum for coordinate system	—	—
3	Determine spindle speeds and feed rates	as desired	107, 116
4	Switch on TNC and machine	—	17
5	Cross over reference marks	—	17
6	Clamp workpiece	—	—
7	Set datum / Reset position display ...		
7a	... **with** the probing functions		33
7b	... **without** the probing functions		31
	Entering and testing part programs		
8	Enter part program or download over external data interface		59
9	Test run: Run part program block by block without tool		103
10	If necessary: Optimize part program		59
	Machining the workpiece		
12	Insert tool and run part program		105

Screen

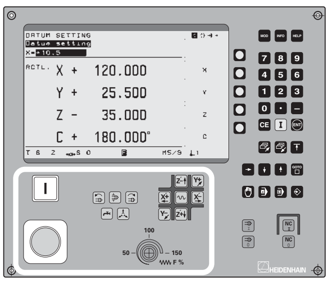


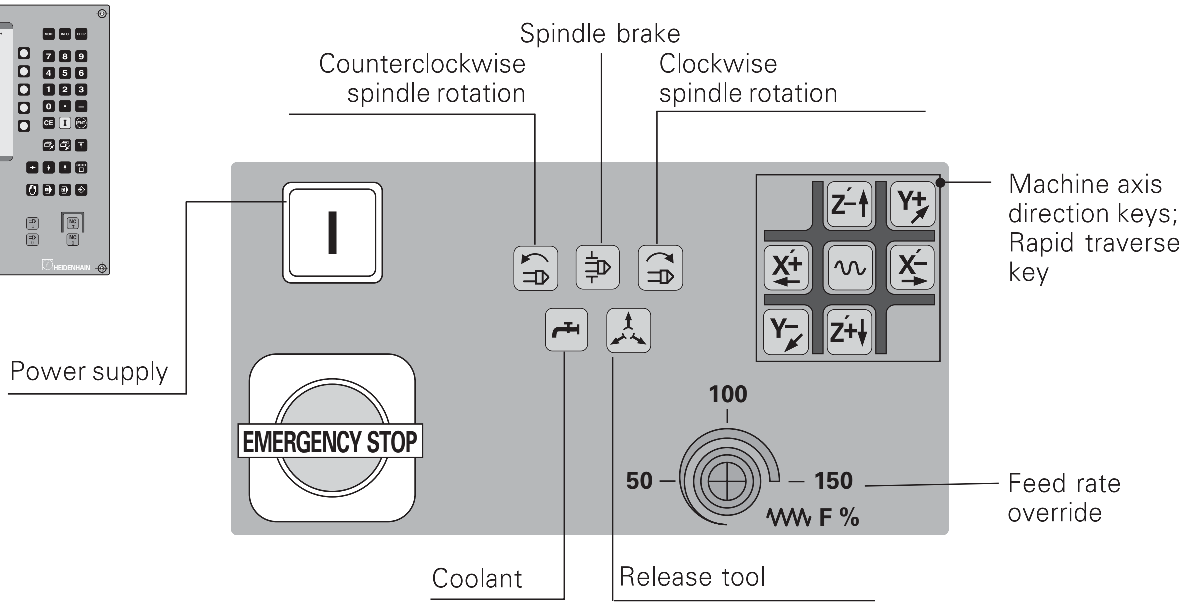


Screen in the operating modes
PROGRAMMING AND EDITING
PROGRAM RUN

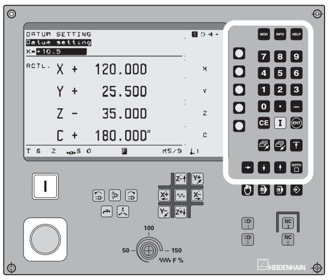


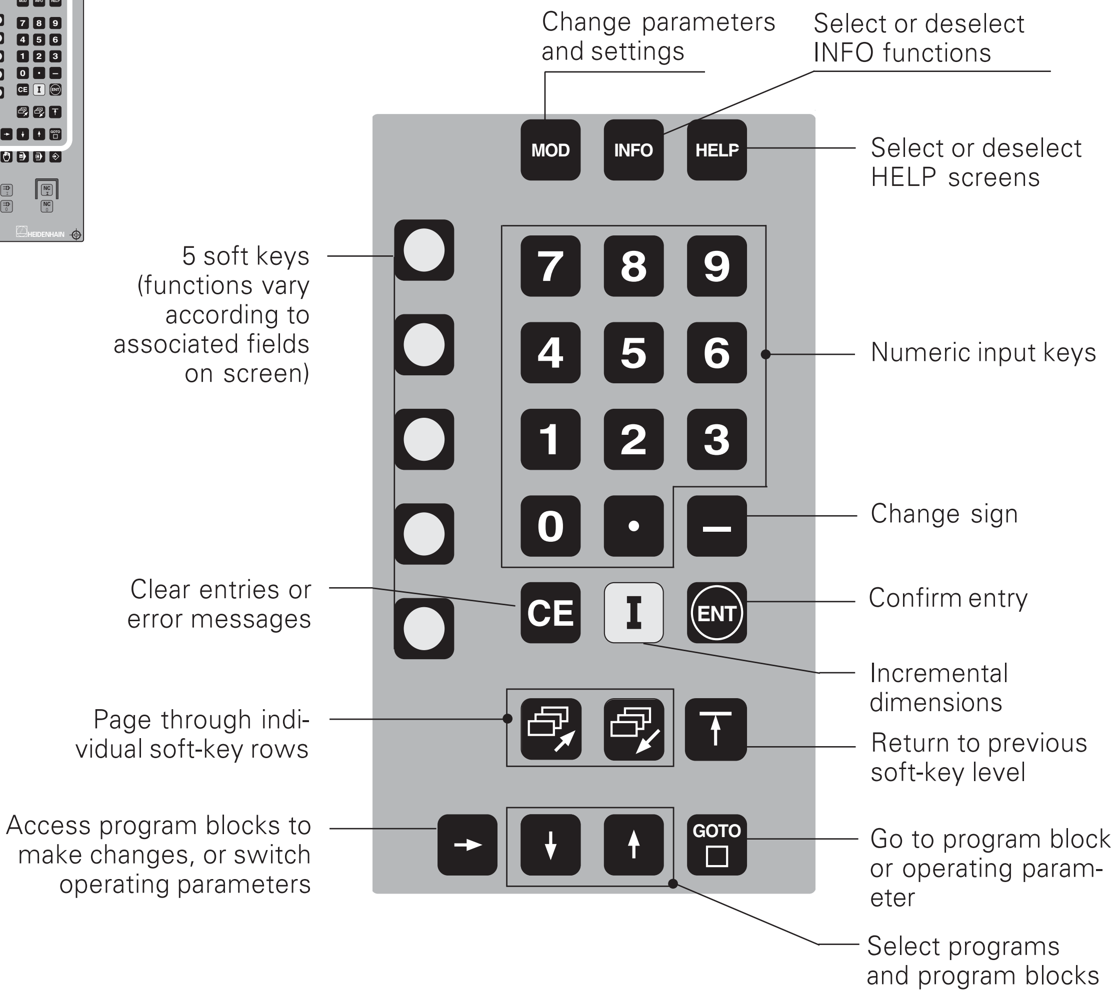
Controlling machine functions



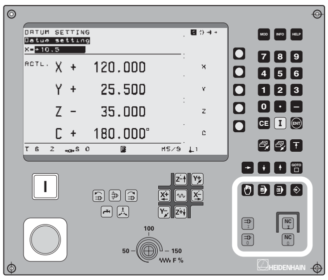


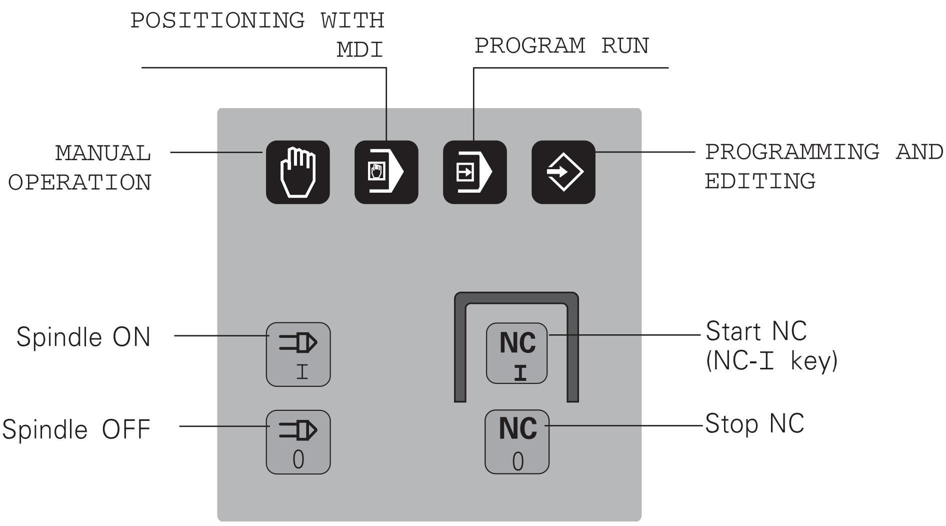
Selecting functions and programming





Selecting operating modes; Start or stop NC and spindle





Contents

Software Version

This User's Manual is for TNC 124 models with the following software version:

Progr. 246 xxx-**16**.

The *x*'s can be any numbers.

For detailed technical information refer to the Technical Manual for the TNC 124.

NC and PLC software numbers

The NC and PLC software numbers of your unit are displayed on the TNC screen after switch-on.

Location of use

The TNC complies with the limits for a Class A device in accordance with the specifications in EN 55022, and is intended for use primarily in industrially-zoned areas.

TNC 124

TNC family

What is NC? NC stands for "**N**umerical **C**ontrol," that is, control of a machine tool by means of numbers. Modern controls such as the TNC have a built-in computer for this purpose and are therefore called CNC (Computerized Numerical Control).

From the very beginning, the TNCs from HEIDENHAIN were developed specifically for shop-floor programming by the machinist. This is why they are called TNC, or **"T**ouch Numerical **C**ontrols."

The **TNC 124** is a straight cut control for boring machines and milling machines with up to three axes. It also features position display of a fourth axis.

Conversational programming

Workpiece machining is defined in a part **program**. It contains a complete list of instructions for machining a part, for example, the target position coordinates, the feed rate and the spindle speed.

You begin programming each machining step by simply pressing a key or soft key. The TNC then asks for all the information that it needs to execute the step.

About This Manual

If you're new to TNC, 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 — so you won't get "lost" too deeply in the theory. As a beginner you should work through all the examples presented.

The **examples** are intentionally brief; it generally won't take you longer than 10 minutes to enter the example data.

If you're already proficient with TNC, you can use the operating instructions as a comprehensive review and reference guide. The clear layout and the subject index make it easy to find the desired topics.

Dialog flowcharts

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

The **operating mode** is indicated above the first dialog flowchart.

This area shows the keys to press.	This area shows the **key function** or **work step**. If necessary, supplementary information will also be included.

Prompt	
This area shows the keys to press.	This area shows 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 **prompt** appears with some actions (not always) at the top of the screen.

If two flowcharts are divided by a **broken line**, and words by **"or,"** this means that you can follow either of the instructions.

Some flowcharts also show the screen that will appear after you press the correct keys.

Abbreviated flowcharts
Abbreviated flowcharts supplement the examples and explanations.
An arrow (➤) indicates a new input or a work step.

Special Notes in this Manual

Particularly important information is presented separately in shaded boxes. Be sure to carefully pay attention to these notes. If you ignore these notes your TNC may not function as required, or damage the workpiece or tool.

Symbols used in the notes

Each note is identified by a symbol to the left. Your manual uses three different symbols which have the following meanings:

General note,
e.g., indicating the behavior of the control.

Note with reference to the **machine manufacturer**,
e.g., indicating that a specific function must be enabled for your machine tool.

Important note,
e.g., indicating that a special tool is required for the function.

TNC Accessories

Electronic handwheel

Electronic handwheels facilitate precise manual control of the axis slides. Like a conventional machine tool, the machine slide moves in direct relation to the rotation of the handwheel. A wide range of traverses per handwheel revolution is available.

The HR 410 Electronic Handwheel

1
Fundamentals of Positioning

Coordinate system and coordinate axes

Reference system

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. The term *coordinate* comes from the Latin word for "that which is arranged." 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 constitutes an absolute reference system.

The Greenwich observatory illustrated in Fig. 1.1 is located at 0° longitude, and the equator at 0° latitude.

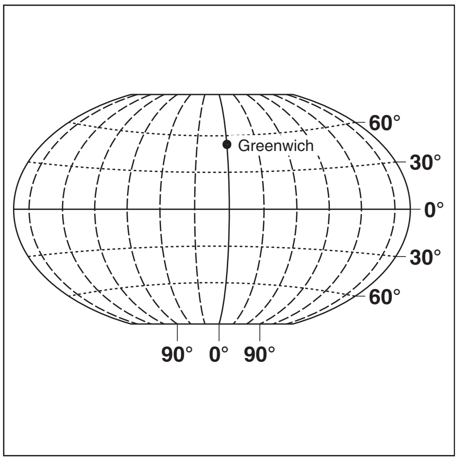


Fig. 1.1: The geographic coordinate system is an absolute reference system

Cartesian coordinate system

On a TNC-controlled milling or drilling machine tool, 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 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.

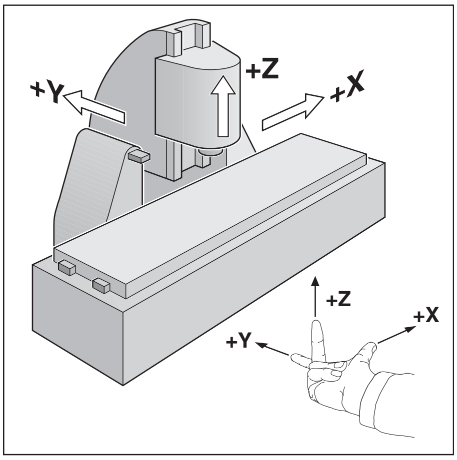


Fig. 1.2: Designations and directions of the axes on a milling machine

Axis designations

X, Y and Z are the main axes of the Cartesian coordinate system. The additional axes U, V and W are secondary linear axes parallel to the main axes. Rotary axes are designated as A, B and C (see Fig. 1.3).

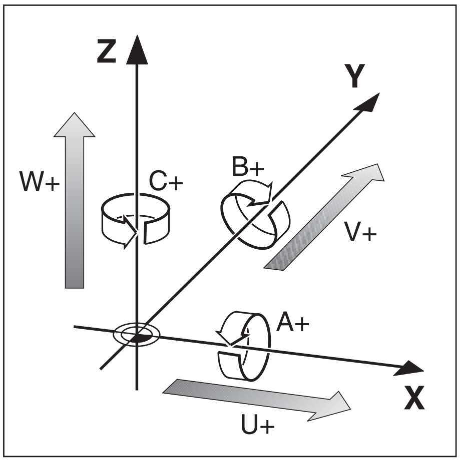


Fig. 1.3: Main, additional and rotary axes in the Cartesian coordinate system

Datums and positions

Setting the datum

The workpiece drawing identifies a certain point on the workpiece (usually a corner) as the “absolute datum” and perhaps one or more other points as relative datums. The datum setting procedure establishes these points as the origin of the absolute or relative coordinate systems: The workpiece, which is aligned with the machine axes, is moved to a certain position relative to the tool and the display is set either to zero or to another appropriate value (e.g., to compensate the tool radius).

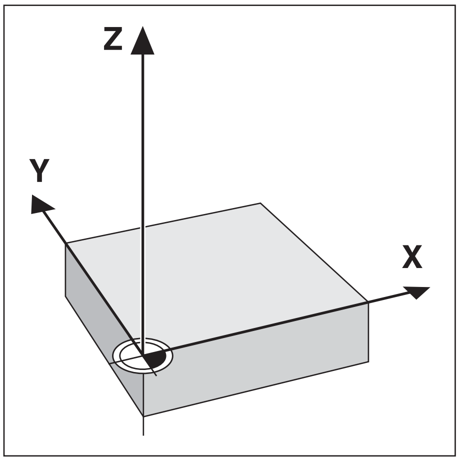


Fig. 1.4: The workpiece datum represents the origin of the Cartesian coordinate system

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 on the X axis and 5 mm from it in the Y axis (in negative direction).

The TNC's probing functions facilitate finding and setting datums.

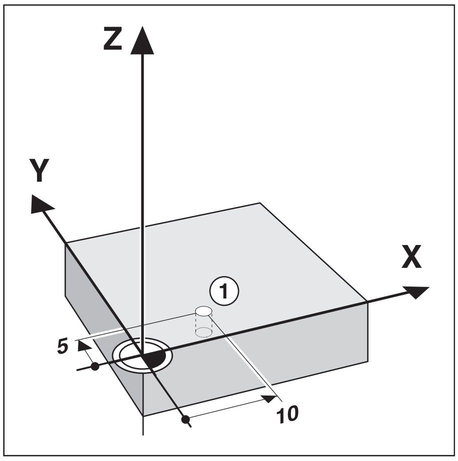


Fig. 1.5: Hole ① defines the coordinate system

Absolute workpiece positions

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

Example: Absolute coordinates of the 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.

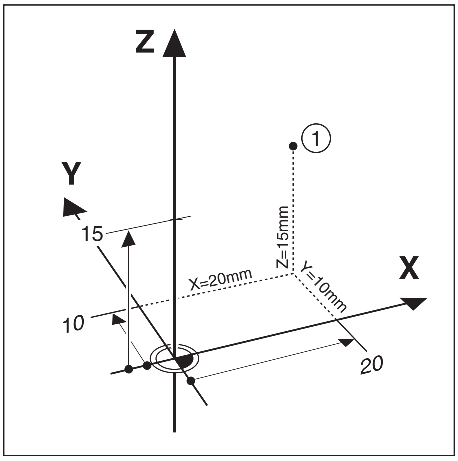


Fig. 1.6: Position definition through absolute coordinates

Incremental workpiece positions

A position can also be referenced to the preceding nominal position. In this case the relative datum is always the last programmed position. Such coordinates are referred to as **incremental coordinates** (increment = increase). They are also called incremental or chain dimensions (since the positions are defined as a chain of dimensions). Incremental coordinates are designated with the prefix **I**.

Example: Incremental coordinates of position ③ referenced to position ②

Absolute coordinates of position ② :

X = 10 mm
Y = 5 mm
Z = 20 mm

Incremental coordinates of position ③ :

IX = 10 mm
IY = 10 mm
IZ = –15 mm

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.

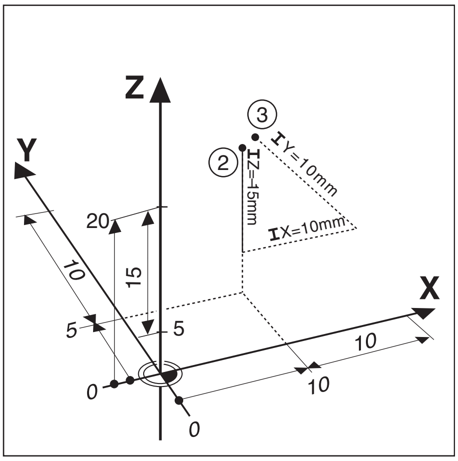


Fig. 1.7: Position definition through incremental coordinates

Machine axis movements and position feedback

Programming tool movements

During workpiece machining, an axis position is changed either by moving the tool or by moving the machine table on which the workpiece is fixed.

> When entering tool movements in a part program you always program as if the tool is moving and the workpiece is stationary.

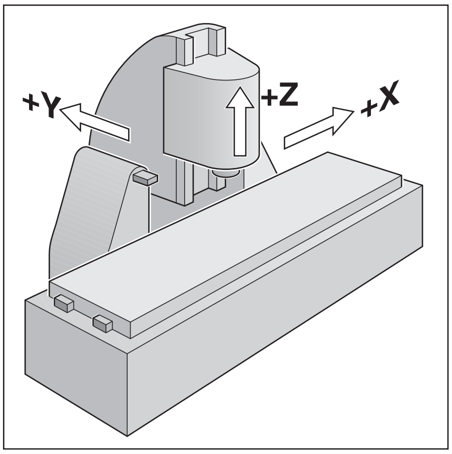


Fig. 1.8: On this machine the tool moves in the Y and Z axes; the workpiece moves in the X axis.

Position feedback

The position feedback encoders — linear encoders for linear axes, angle encoders for rotary axes — convert the movement of the machine axes into electrical signals. The control evaluates these signals and constantly calculates the actual position of the machine axes.

If there is an interruption in power, the calculated position will no longer correspond to the actual position. When power is restored, the TNC can re-establish this relationship.

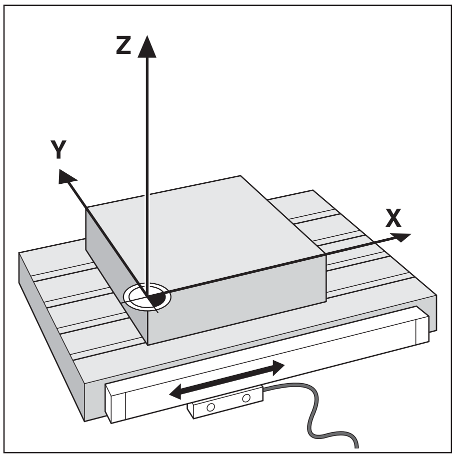


Fig. 1.9: Linear position encoder, here for the X axis

Reference marks

The scales of the position encoders contain one or more reference marks. When a reference mark is passed over, it generates a signal which identifies that position as the reference point (scale reference point = machine reference point). With the aid of this reference mark the TNC can re-establish the assignment of displayed values to machine axis positions.

If the position encoders feature **distance-coded** reference marks, each axis need only move a maximum of 20 mm (0.8 in.) for linear encoders, and 20° for angle encoders.

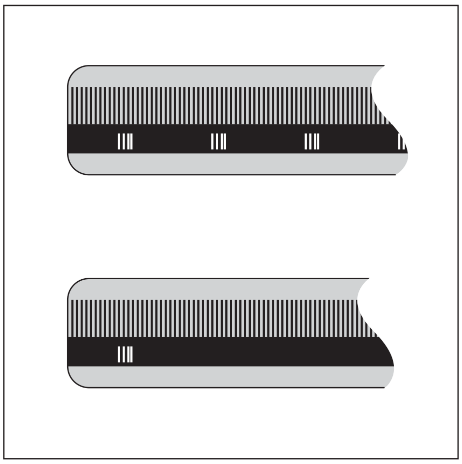


Fig. 1.10: Linear scales: above with distance-coded reference marks, below with one reference mark

Angular positions

For angular positions, the following reference axes are defined:

Plane	Angle reference axis
X / Y	+ X
Y / Z	+ Y
Z / X	+ Z

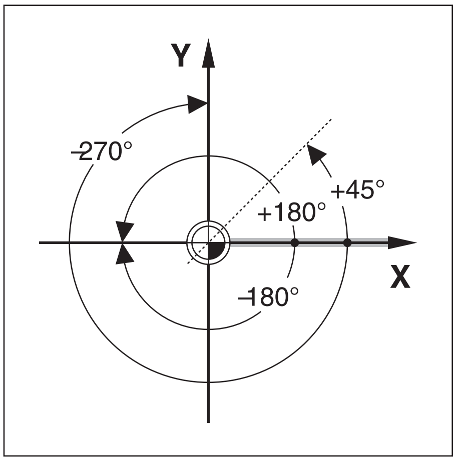


Fig. 1.11: Angle and the angle reference axis, here in the X / Y plane

Algebraic sign for direction of rotation

Positive direction of rotation is counterclockwise if the working plane is viewed in negative tool axis direction (see Fig. 1.11).

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

NOTES

2
Working with the TNC 124 – First Steps

Before you start

You must **cross over the reference marks** after every switch-on. From the positions of the reference marks, the TNC automatically re-establishes the relationship between axis slide positions and display values that you last defined by setting the datum.

Setting up a new datum point automatically stores the new relationship between axis positions and display values.

Switch-on

0 ➤ 1	Switch on the TNC and the machine tool.

MEMORY TEST

Please wait...	The internal memory of the TNC is checked automatically.

POWER INTERRUPTED

CE	Clear the TNC message indicating that the power was interrupted.

RELAY EXT. DC VOLTAGE MISSING

I	Switch on the control voltage. The TNC automatically checks the function of the EMERGENCY STOP button.

CROSS OVER REFERENCE MARKS

For each axis: NC I	Move the axes in the displayed sequence across the reference marks
or	or
Press and hold successively: X+ Y+ Z+	Cross the reference marks in any sequence: Press the machine axis direction button until the moving axis disappears from the screen. Sequence in this example: `X AXIS, Y AXIS, Z AXIS`

The TNC 124 is now ready for operation in the `MANUAL OPERATION` mode.

Operating modes

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

Available functions	Mode	Key
Move the machine axes • with the direction keys, • with the electronic handwheel, • by incremental jog positioning; Datum setting — also with probing functions (e.g. circle center as datum); Enter and change spindle speed and miscellaneous functions	MANUAL OPERATION	
Enter positioning blocks and execute them block by block; Enter hole patterns and execute them block by block; Change spindle speed, feed rate, miscellaneous functions; Enter tool data;	POSITIONING WITH MDI	
Store work steps for small-lot production by • Keyboard entry • Teach-in; Transferring programs through the data interface	PROGRAMMING AND EDITING	
Executing programs • continuously • blockwise	PROGRAM RUN	

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

HELP, MOD and INFO functions

You can call the HELP, MOD and INFO functions at **any time**.

To **call** a function:
➤ Press the function key for that function.

To **leave** a function:
➤ Press the same function key again.

Functions	Designation	Key
On-screen operating instructions: graphics and text explaining the current screen contents	HELP	HELP
User parameters: To redefine the TNC's basic operating characteristics	MOD	MOD
Cutting data calculator, stopwatch, pocket calculator	INFO	INFO

Selecting soft-key functions

The soft-key functions are grouped into one or more rows. The TNC indicates the number of rows by a symbol at the bottom right of the screen.

If no symbol is visible, that means that all pertinent functions are already shown. The highlighted rectangle in the symbol indicates the current row.

Fig. 2.1: The symbol for soft-key rows at the bottom right of the screen. Here, the first row is being displayed.

Overview of functions

Function	Key
Page through the soft-key rows: forwards	
Page through the soft-key rows: backwards	
Go back one soft-key level	

The TNC displays the soft keys with the main functions of an operating mode whenever you press the key for that mode.

Symbols on the TNC screen

The TNC continuously informs you of the current operating status. The symbols are displayed on the screen

- next to the designations of the coordinate axes **or**
- in the status line at the bottom of the screen.

Symbol	Function/Meaning
T ...	Tool, for example T 1
S ... *)	Spindle speed, e.g. S 1000 [rpm]
F ... *)	Feed rate, e.g. F 200 [mm/min]
M ...	Miscellaneous function, e.g. M 3
⊥ ...	Datum, e.g.: ⊥ 1
ACTL.	TNC displays actual values
NOML.	TNC displays nominal values
REF	TNC displays the reference position
LAG	TNC displays the servo lag
✻	Control active
→(O)←	Spindle brake active
←(O)→	Spindle brake inactive
⊗	Axis can be moved with the electronic handwheel

*) A **highlighted F or S symbol** means that the feed rate or spindle has not been enabled by the PLC.

On-screen operating instructions

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

To **call** the operating instructions:

➤ Press the HELP key.

➤ Use the paging keys if the explanation extends over more than one screen page.

To **leave** the operating instructions:

➤ Press the HELP key again.

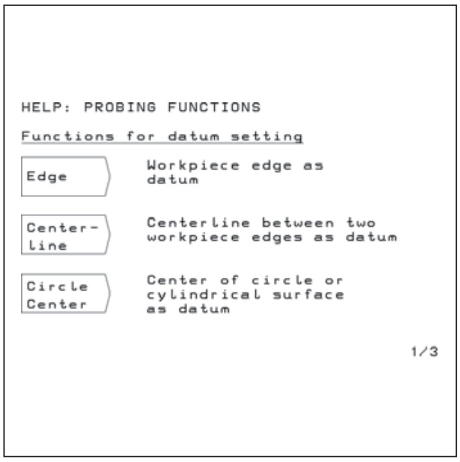


Fig. 2.2: On-screen operating instructions for PROBE, page 1

Example: On-screen operating instructions for datum setting (`PROBE CENTERLINE`)

The `PROBE CENTERLINE` function is described in this manual on page 34.

➤ Select the MANUAL OPERATION mode.

➤ Press the paging key to display the second screen page.

➤ Press the HELP key.
The first page of the operating instructions for the probing functions appears.
Page reference at the lower right of the screen:
the number in front of the slash is the current page, the number behind the slash is the total number of pages.
The on-screen operating instructions now contain the following information on `PROBING FUNCTIONS` (on three pages):
- Overview of the probing functions (page 1)
- Graphic illustration of all probing functions (pages 2 and 3)

➤ To leave the operating instructions:
Press HELP again.
The screen returns to the menu for the probing functions.

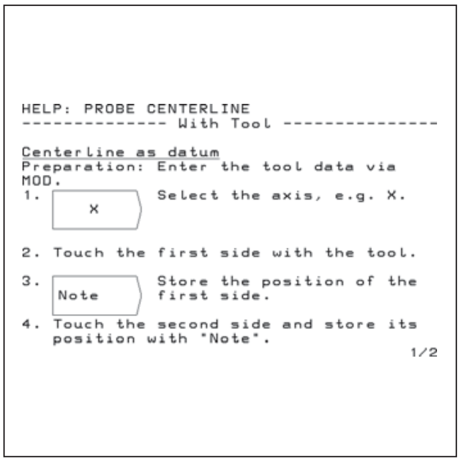


Fig. 2.3: On-screen operating instructions for `PROBE CENTERLINE`, page 1

➤ Press (for example) the soft key `Centerline`.

➤ Press HELP.
The screen now displays operating instructions — spread over three pages — on the function PROBE CENTERLINE including:
- Overview of all work steps (page 1)
- Graphic illustration of the probing sequence (page 2)
- Information on how the TNC reacts and on datum setting (page 3)

➤ To leave the on-screen operating instructions:
Press HELP again.

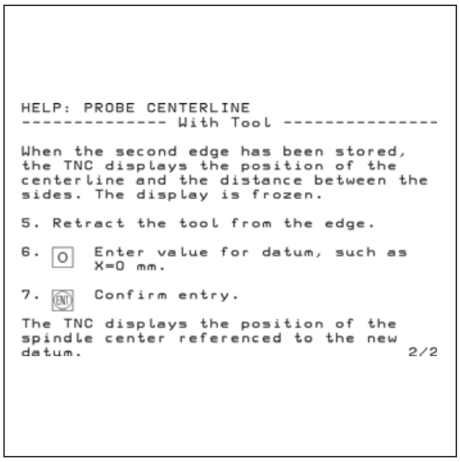


Fig. 2.4: On-screen operating instructions for `PROBE CENTERLINE`, page 2

Error messages

If an error occurs while you are working with the TNC, a message will come up on the screen.

To call an **explanation** of the error:

➤ Press the **HELP** key.

To **clear** the error message:

➤ Press the **CE** key.

Blinking error messages

WARNING!

Blinking error messages mean that the operational reliability of the TNC has been impaired.

If a blinking error message occurs:

➤ Note down the error message displayed on the screen.
➤ Switch off the TNC and machine tool.
➤ Attempt to correct the problem with the power off.
➤ If the error cannot be corrected or if the blinking error message recurs, notify your customer service agency.

Selecting the unit of measurement

Positions can be displayed in millimeters or inches. If you choose inches, `inch` will be displayed at the top of the screen.

To **change** the unit of measurement:

➤ Press MOD.
➤ Page to the soft-key row containing the user parameter `mm` or `inch`.
➤ Choose the soft key `mm` or `inch` to change to the other unit.
➤ Press MOD again.

For more information on user parameters, see Chapter 13.

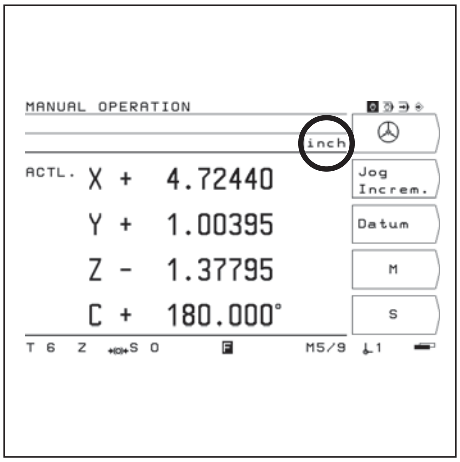


Fig. 2.5: The `inch` indicator

Selecting position display types

The TNC can display various position values for a specific tool position.

The positions indicated in Fig. 2.6 are:

- Starting position of the tool Ⓐ
- Target position of the tool Ⓩ
- Workpiece datum Ⓦ
- Scale reference point Ⓜ

The TNC position display can be set to show the following types of information:

- Nominal position `NOML.` ①
  The value presently commanded by the TNC.
- Actual position `ACTL.` ②
  The position at which the tool is presently located as referenced to the workpiece datum.
- Servo lag `LAG` ③
  The difference between nominal and actual positions (`NOML. - ACTL.`)
- Actual position as referenced to the scale reference point `REF` ④

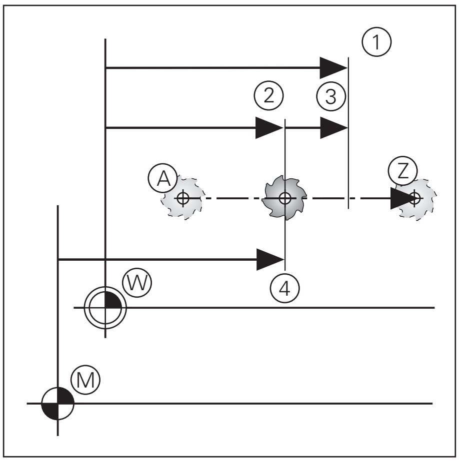


Fig. 2.6: Tool and workpiece positions

To change the position display

➤ Press MOD.
➤ Page to the soft-key row containing the user parameter `Posit`.
➤ Press the soft key for selecting the position display type and change to the other display type.
➤ Select the desired display type.
➤ Press MOD again.

For more information on user parameters, see Chapter 13.

Traverse limits

The maximum range of traverse of the machine axes is set by the machine manufacturer.

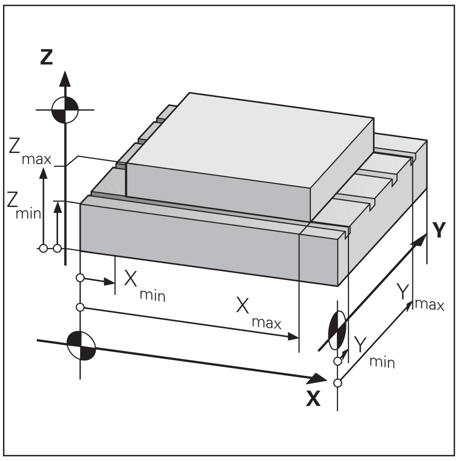


Fig. 2.7: Traverse limits define the machine's actual working envelope

3
Manual Operation and Setup

> The machine manufacturer may define a method of moving the axes that varies from what is described in this manual.

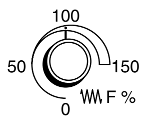
On the TNC 124 you can move the machine axes with:
- the direction keys,
- the electronic handwheel,
- incremental jog positioning, or
- positioning with manual data input MDI (see Chapter 4).

In the `MANUAL OPERATION` and `POSITIONING WITH MDI` modes of operation (see Chapter 4) you can also enter and change:
- Feed rate F (the feed rate can only be entered in `POSITIONING WITH MDI`)
- Spindle speed S
- Miscellaneous function M

Feed rate F, spindle speed S and miscellaneous function M

To change the feed rate F:

You can vary the feed rate F infinitely by turning the knob for feed rate override on the TNC control panel.

Feed rate override	
You can vary the feed rate F from 0% to 150% of the set value	

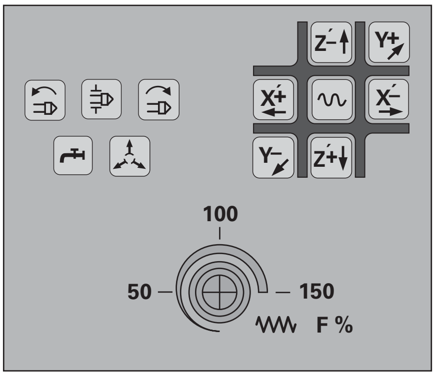


Fig. 3.1: Feed rate override on the TNC control panel

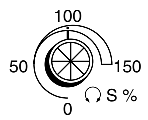
Entering and changing the spindle speed S

> The machine manufacturer determines which spindle speeds are allowed on your TNC.

Example: Entering the spindle speed S

S	Select S for the spindle speed function.
Spindle speed ?	
9 5 0	Enter the spindle speed, for example 950 rpm.
NC I	Change the spindle speed.

To change the spindle speed S:
You can vary the spindle speed S infinitely by turning the knob for spindle speed override—if provided—on the TNC control panel.

Spindle speed override	
You can vary the spindle speed S from 0% to 150% of the set value	

Entering a miscellaneous function M

> The machine manufacturer determines which miscellaneous functions are available on your TNC and which effects they have.

Example: Entering a miscellaneous function

M	Select M for miscellaneous function.
Miscellaneous function M ?	
3	Enter the miscellaneous function, for example M 3: spindle ON, clockwise.
NC I	Execute the miscellaneous function.

Moving the machine axes

The TNC control panel includes six direction keys. The keys for the X and Y axes are identified with a prime mark (X', Y'). This means that the traversing directions indicated on these keys correspond to movement of the machine table.

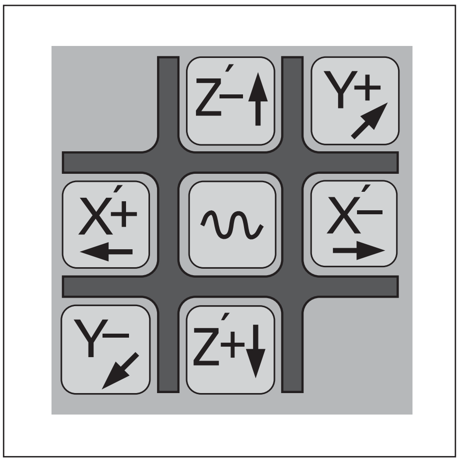


Fig. 3.2: The direction keys on the TNC control panel, with the key for rapid traverse in the center

Traversing with the direction keys

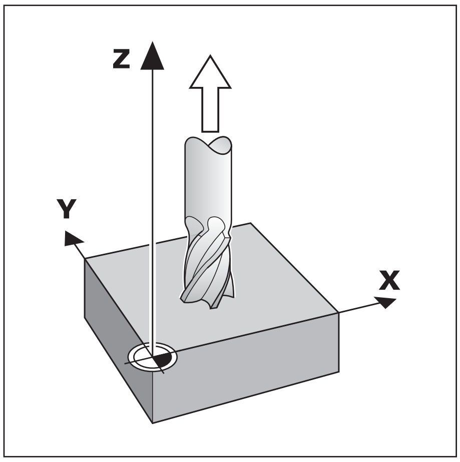
The direction key defines at the same time
- the coordinate axis, for example **X**
- the traversing direction, for example negative: X**–**

On machine tools with **central drives** you can only move one axis at a time.

If you are moving a machine axis with the direction key, the TNC automatically stops moving the axis as soon as you release the key.

For continuous movement:
You can also move the machine axes continuously:
The axis continues to move after you release the key.
To stop the axis press the key indicated below in example 2.

Rapid traverse
To move an axis at rapid traverse:
➤ Press the rapid traverse key and the direction key together.



Example: Moving the machine axis in the Z+ direction with the direction key (retract tool):

Example 1: Moving the machine axes

Operating mode: `MANUAL OPERATION`

Press and hold: Z'+	Press the direction key, here for the positive Z direction (Z'+) and hold it as long as you wish the axis to move.

Example 2: For continuous movement of the machine axes

Operating mode: `MANUAL OPERATION`

Together: Z'+ NC I	Start movement of the axis: Press the direction key, here for the positive Z direction (Z'+) together with the NC-`I` key.
NC 0	Stop the axis.

Traversing with the electronic handwheel

> Electronic handwheels can be connected only to machines with preloaded drives. The machine manufacturer can tell you whether electronic handwheels can be connected on your machine.

You can connect the following HEIDENHAIN electronic handwheels to your TNC 124:

- HR 410 portable handwheel
- HR 130 integral handwheel

Direction of traverse
The machine manufacturer determines in which direction the handwheel must be turned to move an axis in a specific direction.

If you are working with the HR 410 portable handwheel
The HR 410 portable handwheel is equipped with two permissive buttons ③. You can move the machine axes with the handwheel ② only if a permissive button is depressed.

Other features of the HR 410:

- Axis selection keys X, Y and Z ④ .
- The axes can be moved continuously with the + and – direction keys ⑦ .
- Three keys for slow, medium and fast traverse ⑥ .
- Actual-position-capture key ⑤ for transferring positions or tool data in teach-in mode directly from the position display into the program or tool table (without having to type the numbers).
- Three keys for machine functions ⑧ defined by the machine tool builder.
- EMERGENCY STOP button ① for immediate machine shutdown in case of danger. This safety feature is additional to the permissive buttons.
- Magnetic holding pads on the back of the handwheel enable you to place it within easy reach on a flat metal surface.

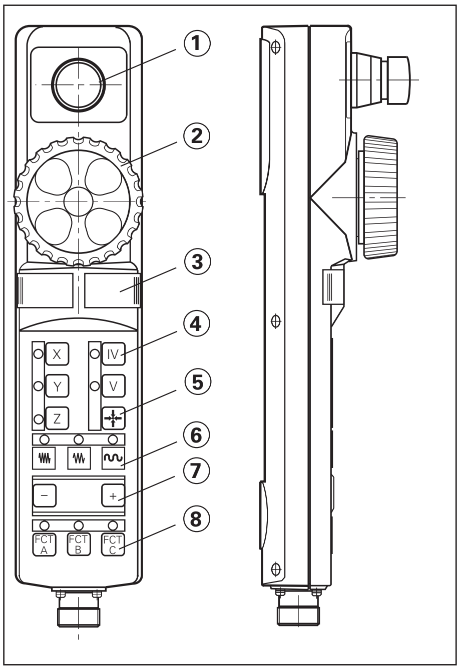


Fig. 3.3: The HR 410 portable electronic handwheel

Example: Moving a machine axis with the HR 410 electronic handwheel, for example the Y axis

Operating mode: `MANUAL MODE`

	Select the `Electronic handwheel` function. The handwheel symbol is displayed next to the “X” for the X coordinate.
Y	Select the coordinate axis at the handwheel. The handwheel symbol is shifted to the selected coordinate axis.
	Select the traverse per revolution: large, medium, or small, as preset by the machine tool builder.
	Press the permissive button! Turn the handwheel to move the machine axis.

Incremental jog positioning

Incremental jog positioning enables you to move a machine axis by the increment you have preset each time you press the corresponding direction key.

Current jog increment
If you enter a jog increment, the TNC stores the entered value and displays it right of the highlighted input line for `Infeed`.
The programmed jog increment is effective until a new value is entered by keyboard or soft key.

Maximum input value
0.001 mm ≤ jog increment ≤ 99.999 mm

Changing the feed rate F
You can increase or decrease the feed rate F by turning the knob for feed rate override.

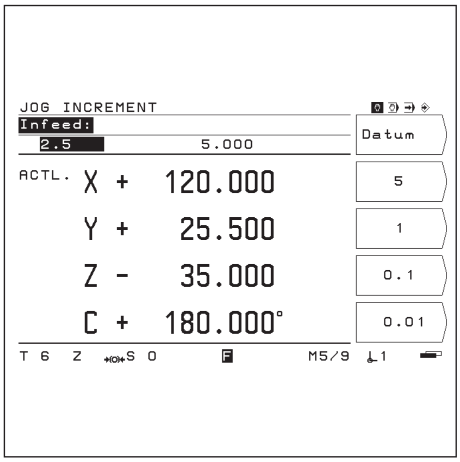
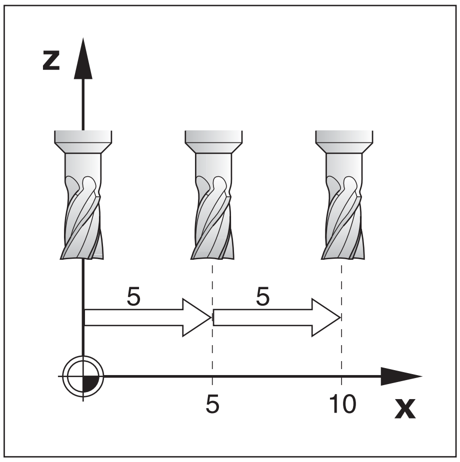


Fig. 3.4: TNC screen for incremental jog positioning

Example: Moving the machine axis in the X+ direction by incremental jog positioning



Operating mode: `MANUAL OPERATION`

Jog Increm. (soft key)	Select the `Jog Increm.` function.
`Infeed : 0.000`	
5 (soft key)	Enter the infeed (5 mm)—by soft key.
or	or
5 ENT	Enter the infeed (5 mm)—with the keyboard and confirm your entry with ENT.
`Infeed : 0.000 5.000`	
X+	Move the machine axis by the entered infeed, for example in the X+ direction.

Entering tool length and radius

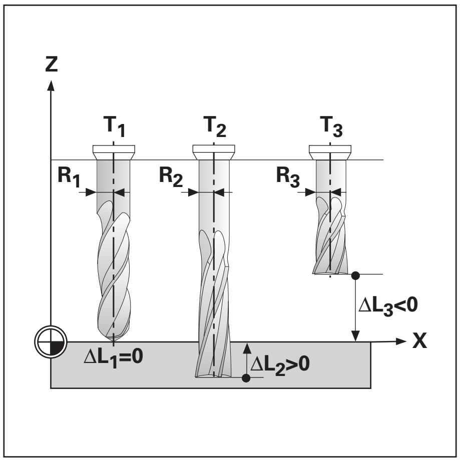


Fig. 3.5: Tool length and radius

Enter the lengths and radii of your tools in the TNC's tool table. The TNC will then take the entered data into account for datum setting and all other machining processes.

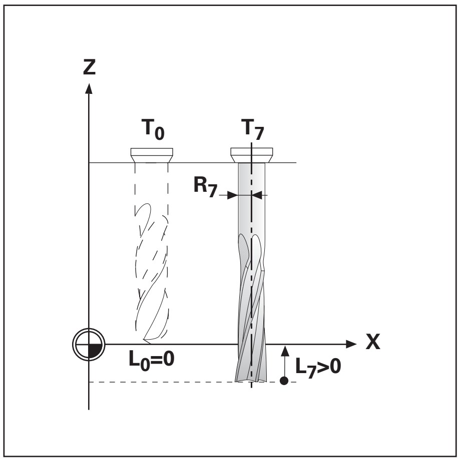
You can enter up to 99 tools.

The tool length is the difference in length ΔL between the tool and the zero tool.

To enter the tool length directly move the tool until it touches the workpiece and transfer the tool position coordinate by using the "actual position capture" function.

Sign for the length difference ΔL
If the tool is **longer** than the zero tool: $\Delta L > 0$
If the tool is **shorter** than the zero tool: $\Delta L < 0$

Example: Entering the tool length and radius into the tool table



Tool number:	e.g. 7
Tool length:	L = 12 mm
Tool radius:	R = 8 mm

MOD	Select the user parameters.
/	Go to the soft-key row containing `Tool Table`.
Tool Table	Open the tool table.

Tool number ?

7 ENT	Enter the tool number (such as 7) and confirm your entry with ENT.

Tool length ?

1 2 ENT	Enter the tool length (12 mm) and confirm your entry with ENT.
or	or
	Capture the actual position in the tool axis by pressing the soft key.
or	or
	Capture the actual position in the tool axis by pressing key on the handwheel.

Tool radius ?

8 ENT	Enter the tool radius (8 mm) and confirm your entry with ENT.
MOD	Depart the user parameters.

Calling the tool data

The lengths and radii of your tools must first be entered into the TNC's tool table (see previous page).

Before you start workpiece machining, select the tool you are using from the tool table. To call the desired tool, move the highlight to the tool, select the axis with the corresponding soft key and press the soft key `Tool Table`.

The TNC 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.

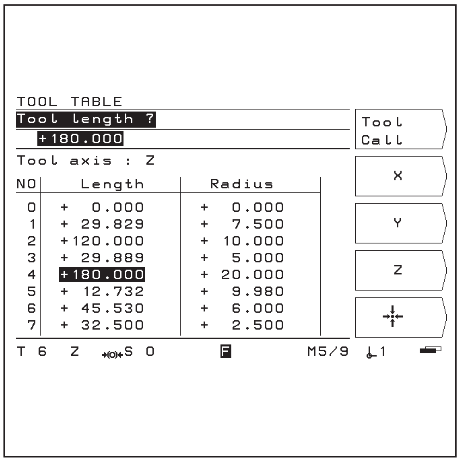


Fig. 3.6: The tool table on the TNC screen

Example: Calling the tool data

MOD	Select the user parameters.
/	Go to the first soft-key row containing `Tool Table`.
Tool Table	Select the tool table.

Tool number ?

5 ENT	Enter the tool number (here: 5) and confirm your entry with ENT.
Z	Select the `Tool axis` (Z).
Tool Call	"Activate" the tool and depart the user parameters.

Selecting datum points

The TNC 124 can store up to 99 datum points in a datum table. In most cases this will free you from having to calculate the axis travel when working with complicated workpiece drawings containing several datums, or when several workpieces are clamped to the machine table at the same time.

For each datum point, the datum table contains the positions that the TNC 124 assigned to the reference point on the scale of each axis (REF values) during datum setting. Note that if you change the REF values in the table, this will move the datum point.

The TNC 124 displays the number of the current datum at the lower right of the screen.

To select the datum:

In all operating modes:

- Press MOD and go to the soft key row containing `Datum Table`.
- Choose the soft key `Datum Table`.
- Select the datum you are using from the datum table.
- Leave the datum table:
  Press MOD again.

In the MANUAL OPERATION and POSITIONING WITH MDI modes of operation:

- Press the vertical arrow keys.

> The machine manufacturer determines whether "quick datum selection" via arrow keys is enabled on your TNC.

In the PROGRAMMING AND EDITING / PROGRAM RUN modes of operation:

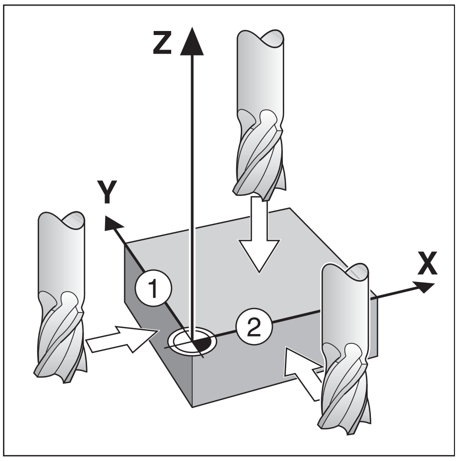
- You can also select a datum point by entering the command "DATUM" in a program.

Datum setting: Approaching positions and entering actual values

The easiest way to set datum points is to use the TNC's probing functions. A description of the probing functions starts on page 33.

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 on this page and the next).

Example: Setting a workpiece datum without the probing function



Working plane:	X / Y
Tool axis:	Z
Tool radius:	R = 5 mm
Axis sequence in this example:	X - Y - Z

Preparation

- ➤ Select the desired datum point (see "Selecting datum points").
- ➤ Insert the tool.
- ➤ Press MOD and go to the soft-key row containing `Tool Table`.
- ➤ Select the user parameter `Tool Table`.
- ➤ Select the tool you will use to set the datum.
- ➤ Leave the tool table: Press the soft key `Tool Call`.
- ➤ Activate the spindle, for example with the miscellaneous function M 3.

Operating mode: `MANUAL OPERATION`

Datum	Select the `Datum` function.
X	Select the X axis.
	Touch edge ① with the tool.
Datum setting **X = + 0**	
– 5 ENT	Enter the position of the tool center (X = –5 mm) and transfer the X coordinate of the datum.
Y	Select the Y axis.
	Touch edge ② with the tool.
Datum setting **Y = – 5**	
ENT	Transfer the Y coordinate of the datum.
Z	Select the Z axis.
	Touch the workpiece surface.
Datum setting **Z = – 5**	
0 ENT	Enter the position of the tool tip (Z = 0 mm) and transfer the Z coordinate of the datum.

Functions for datum setting

It is very easy to set datum points with the TNC's probing functions. These functions do not require a touch probe system or an edge finder since you simply probe the workpiece edges with the tool.

The following probing functions are available:

- Workpiece edge as datum:
  `Edge`
- Centerline between two workpiece edges:
  `Centerline`
- 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.

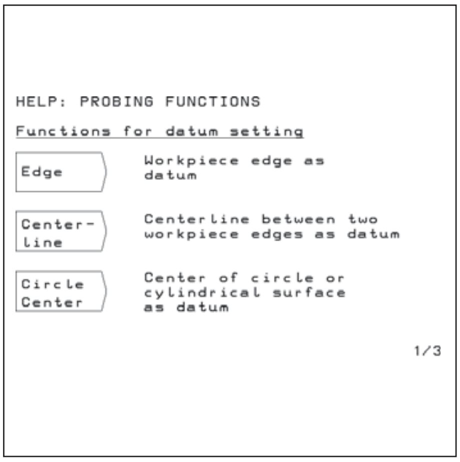


Fig. 3.7: On-screen operating instructions for the probing functions

Preparations for all probing functions

➤ Select the desired datum point (see “Selecting datum points”).
➤ Insert the tool.
➤ Press MOD and go to the soft-key row containing `Tool Table`.
➤ Select the user parameter `Tool Table`.
➤ Select the tool you will use to set the datum.
➤ Leave the tool table:
Press the soft key `Tool Call`.
➤ Activate the spindle, for example with the miscellaneous function M3.

To abort the probing function

While the probing function is active, the TNC displays the soft key `Escape`. Choose this soft key to return to the opening state of the selected probing function.

Measuring diameters and distances

With the probing function `Centerline` the TNC calculates the distance between the two probed edges of a workpiece; with the `Circle Center` function it determines the diameter of the probed circle.

The calculated distance and diameter are displayed on the TNC screen between the position displays.

If you want to measure the distance between two edges or a diameter **without** setting a datum:

➤ Probe the workpiece as described on page 35 (`Centerline`) and page 36 (`Circle Center`).

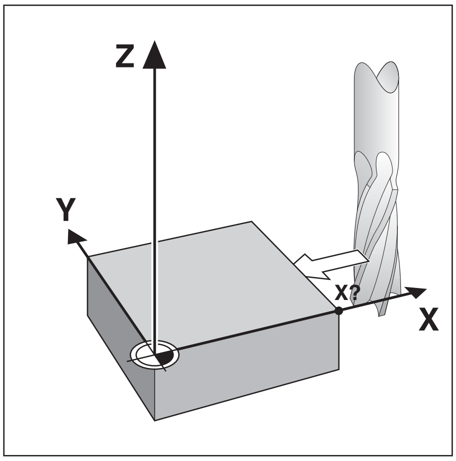
As soon as the TNC displays the distance or diameter:

➤ **Do not** enter a datum coordinate. Simply press the soft key `Escape`.

Example: Probe workpiece edge, display position of workpiece edge and set the edge as a datum

The probed edge lies parallel to the Y axis.

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



Operating modes: `MANUAL OPERATION/ELECTRONIC HANDWHEEL/JOG INCREMENT`

/	Go to the second soft-key row.
`Edge`	Select `Edge`.
`X`	Select the axis for which the coordinate is to be set: X axis.

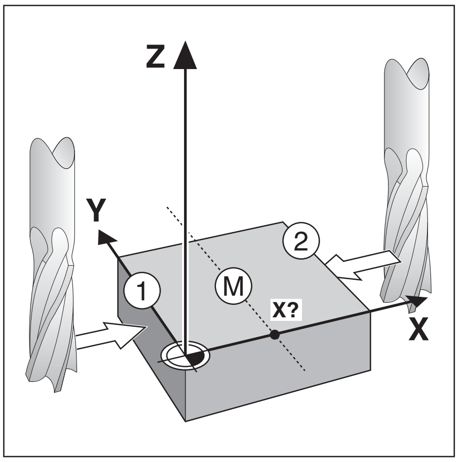
`Probe in X axis`

	Move the tool towards the workpiece until it makes contact.
`Note`	Store the position of the workpiece edge.
	Retract the tool from the workpiece.

`Enter value for X`
`+ 0`

2 0 ENT	0 is offered as a default value for the coordinate. Enter the desired coordinate for the workpiece edge, for example X = 20 mm and set the coordinate as a datum for this workpiece edge.

Example: Set centerline between two workpiece edges as datum



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

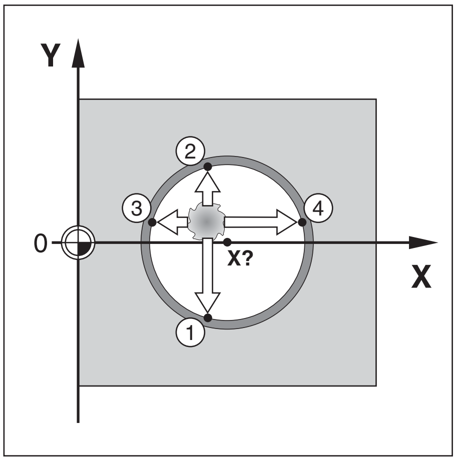
The centerline is parallel to the Y axis.

Desired coordinate
of the centerline: X = 5 mm

Operating modes: MANUAL OPERATION/ELECTRONIC HANDWHEEL/JOG INCREMENT

/	Go to the second soft-key row.
Center-line	Select Centerline.
X	Select the axis for which the coordinate is to be set: X axis.
Probe 1st edge in X	
	Move the tool towards workpiece edge ① until it makes contact.
Note	Store the position of the edge.
Probe 2nd edge in X	
	Move the tool towards workpiece edge ② until it makes contact.
Note	Store the position of the edge. Screen display is frozen; the distance between the two edges is displayed below the selected axis.
	Retract the tool from the workpiece.
Enter value for X **+ 0**	
5 ENT	Enter coordinate (X = 5 mm) and transfer coordinate as datum for the centerline.

Example: Probe the circumference of a hole and set the center of the hole as a datum



Main plane:	X / Y plane
Tool axis:	Z
X coordinate of the circle center:	X = 50 mm
Y coordinate of circle center:	Y = 0 mm

Operating mode: `MANUAL OPERATION/ELECTRONIC HANDWHEEL/JOG INCREMENT`

/	Go to the second soft-key row.
Circle Center	Select `Circle Center`.
Plane X/Y	Select plane containing the circle (main plane): `Plane X/Y`.
Probe 1st point in X / Y	
	Move tool towards first point 1 on the circumference until it makes contact.
Note	Store position of the bore hole wall.
	Retract tool from bore hole wall.
	Probe three additional points on the circumference in the same manner. Further information appears on the screen. Store positions with `Note`.
Enter center point X **X = 0**	
5 0 ENT	Enter first coordinate (X = 50 mm) and transfer coordinate as datum for the circle center.
Enter center point Y **Y = 0**	
ENT	Accept default entry Y = 0 mm.

NOTES

4
Positioning with Manual Data Input (MDI)

For many simple machining processes, for example if a machining process is to be executed only once, or if you are machining simple geometrical shapes, it would be too time-consuming to enter the individual machining steps in an NC program.

In the `POSITIONING WITH MDI` mode of operation you can enter all data directly instead of storing them in a part program.

Simple milling and drilling operations
Enter the following nominal position data manually in the `POSITIONING WITH MDI` mode of operation:
- Coordinate axis
- Position value
- Radius compensation

The TNC then moves the tool to the desired position.

Pecking and tapping, hole patterns, rectangular pocket milling
The `POSITIONING WITH MDI` mode of operation also supports the TNC "Cycles" (see Chapter 7):
- Pecking
- Tapping
- Bolt hole circle patterns
- Linear hole patterns
- Rectangular pocket

Before you machine the workpiece

➤ Select the desired datum point (see "Selecting datum points").
➤ Insert the tool.
➤ Pre-position the tool to prevent the possibility of damaging the tool or workpiece.
➤ Select an appropriate feed rate F.
➤ Select an appropriate spindle speed S.

Taking the tool radius into account

The TNC can compensate for the tool radius (see Fig. 4.1).
This allows you to enter workpiece dimensions directly from the drawing. The remaining distance is then automatically lengthened (R+) or shortened (R–) by the tool radius.

Entering tool data

➤ Press MOD.
➤ Choose the soft key `Tool Table`.
➤ Enter the tool number.
➤ Enter the tool length.
➤ Enter the tool radius.
➤ Select the tool axis via soft key.
➤ Press the `Tool Call` soft key.

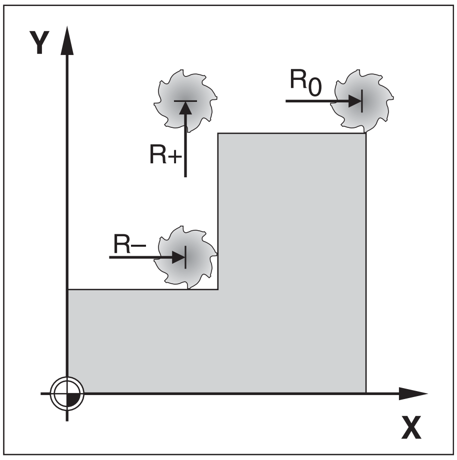


Fig. 4.1: Tool radius compensation

Feed rate F, spindle speed S and miscellaneous function M

In the `POSITIONING WITH MDI` mode of operation you can also enter and change the following information:

- Feed rate F
- Spindle speed S
- Miscellaneous function M

Feed rate F after an interruption of power

If you have entered a feed rate F in the `POSITIONING WITH MDI` mode of operation, the TNC will move the axes with this feed rate after an interruption of power as soon as power is restored.

Entering and changing the feed rate F

Example: Entering the feed rate F

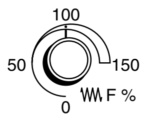
F	Select `F` for the feed rate function.
Feed rate ?	
5 0 0	Enter the feed rate F, for example 500 mm/min.
ENT	Confirm the feed rate F for the next positioning step.

Changing the feed rate F

You can vary the feed rate F infinitely by turning the knob for feed rate override on the TNC control panel.

Feed rate override

You can vary the feed rate F from 0% to 150 % of the entered value.



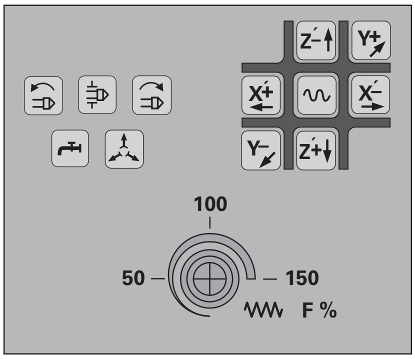


Fig. 4.2: Knob for feed rate override on the TNC control panel

Entering and changing the spindle speed S

> The machine manufacturer determines which spindle speeds are allowed on your TNC.

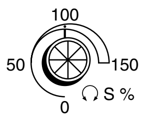
Example: Entering the spindle speed S

S	Select S for the spindle speed function.
Spindle speed ?	
9 5 0	Enter the spindle speed S, for example 950 rpm.
NC I	Change the spindle speed S.

To change the spindle speed S:
You can vary the spindle speed S infinitely by turning the knob for spindle speed override—if provided—on the TNC control panel.

Spindle speed override

You can vary the spindle speed S from 0% to 150% of the set value.



Entering a miscellaneous function M

> The machine manufacturer determines which miscellaneous functions are available on your TNC and what effects they have.

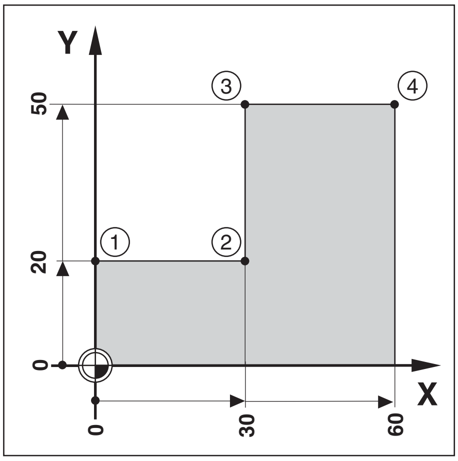
Example: Entering a miscellaneous function

M	Select M for the miscellaneous functions.
Miscellaneous function M ?	
3	Enter the miscellaneous function M, for example M 3: spindle ON, clockwise.
NC I	Execute the miscellaneous function M.

Entering and moving to positions

For simple machining operations, you can program the coordinates directly in the `POSITIONING WITH MDI` mode of operation.

Example: Milling a shoulder



The coordinates are entered as absolute dimensions; the datum is the workpiece zero.

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

Preparation:

➤ Select the desired datum point (see "Selecting datum points").

➤ Enter the tool data.

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

➤ Move the tool to milling depth.

Operating mode: `POSITIONING WITH MDI`

Y	Select the Y axis.
Nominal position value ?	
2 0 / Radius Comp.	Enter the nominal position value for corner point ①: Y = + 20 mm and select tool radius compensation: R +.
NC I	Move the tool to the programmed position.
X	Select the X axis.
Nominal position value ?	
3 0 / Radius Comp.	Enter the nominal position value for corner point ②: X = + 30 mm and select tool radius compensation: R –.
NC I	Move the tool to the programmed position.

Y	Select the Y axis.

Nominal position value ?

5 0 Radius Comp.	Enter the nominal position value for corner point ③: Y = + 50 mm and select tool radius compensation: R +.
NC I	Move the tool to the programmed position.
X	Select the X axis.

Nominal position value ?

6 0	Enter the nominal position value for corner point ④: X = + 60 mm, tool radius compensation is already set to R +.
NC I	Move the tool to the programmed position.

Pecking and tapping

The TNC cycles for pecking and tapping (see Chapter 7) are available in the `POSITIONING WITH MDI` mode of operation.

Use the soft keys on the second soft-key row to select the desired type of hole and enter the required data. These data can usually be taken from the workpiece drawing (hole depth, infeed depth, etc.).

The TNC controls the machine tool and calculates additional data such as the advanced stop distance if the hole is to be drilled in several infeeds.

Pecking and tapping in hole patterns

The functions for pecking and tapping are also available in combination with the hole pattern functions Circle Pattern and Linear Pattern.

Pecking and tapping processes

The input data for pecking and tapping can also be entered as "cycles" in a part program. You will find detailed information on how the TNC controls pecking and tapping operations in Chapter 7. (See page 79 for pecking and page 82 for tapping).

Pre-positioning the drill for pecking and tapping

Pre-position the drill in the Z axis to a position above the workpiece.

In the X and Y axes (working plane), pre-position the drill to the hole position. The hole position is approached without radius compensation (input R0).

Input data for pecking

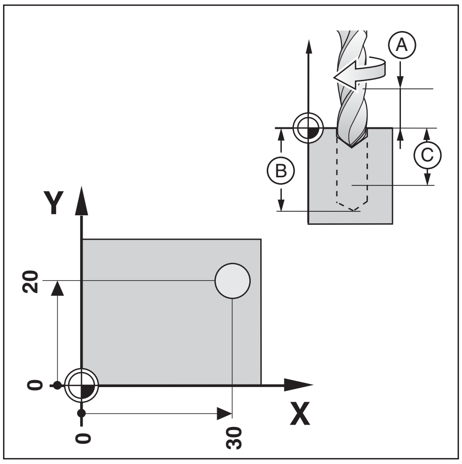
- Clearance height at which the drill can traverse in the working plane without damaging the workpiece;
  Enter an absolute value together with the algebraic sign.
- Setup clearance at which the drill is located above the workpiece.
- Coordinate of the workpiece surface;
  Enter an absolute value together with the algebraic sign.
- Hole depth; the algebraic sign determines the working direction.
- Infeed depth
- Dwell time of the drill at the bottom of the hole.
- Machining feed rate

Input data for tapping

- Clearance height at which the drill can traverse in the working plane without damaging the workpiece;
  Enter an absolute value together with the algebraic sign.
- Setup clearance at which the drill is located above the workpiece.
- Coordinate of the workpiece surface;
  Enter an absolute value together with the algebraic sign.
- Hole depth; the algebraic sign determines the working direction.
- Dwell time of the drill at the end of thread.
- Machining feed rate

Example: PECKING

X coordinate of the hole:	30	mm
Y coordinate of the hole:	20	mm
Clearance height:	+ 50	mm
Setup clearance Ⓐ :	2	mm
Workpiece surface:	+ 0	mm
Hole depth Ⓑ :	– 15	mm
Pecking depth Ⓒ :	5	mm
Dwell time:	0.5	s
Pecking feed rate:	80	mm/min
Hole diameter:	e.g. 6	mm



Preparation

➤ Pre-position the tool over the workpiece.

Operating mode: POSITIONING WITH MDI

X	Select the X axis.

Nominal position value ?

3 0 Radius Comp.	Enter the nominal position value for pre-positioning in the X axis: X = + 30 mm and select tool radius compensation R 0.
NC I	Pre-position the tool in the X axis.
Y	Select the Y axis.

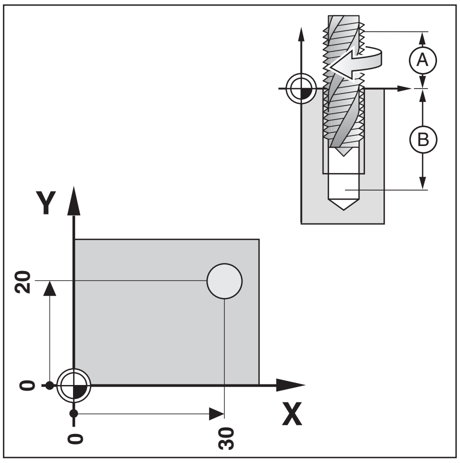
Nominal position value ?

2 0	Enter the nominal position value for pre-positioning in the Y axis: Y = + 20 mm. Tool radius compensation is already set to R 0.
NC I	Pre-position the tool in the Y axis.

/	Go to the second soft-key row.
Pecking	Select Pecking.
Clearance height ?	
5 0 ENT	Enter the clearance height of the tool over the workpiece (+ 50 mm). Confirm your entry.
Setup clearance ?	
2 ENT	Enter setup clearance Ⓐ (2 mm). Confirm your entry.
Surface ?	
0 ENT	Enter the coordinate of the workpiece surface (0 mm). Confirm your entry.
Hole depth ?	
– 1 5 ENT	Enter hole depth Ⓑ (– 15 mm). Confirm your entry.
Pecking depth ?	
5 ENT	Enter pecking depth C (5 mm). Confirm your entry.
Dwell time ?	
0 • 5 ENT	Enter the dwell time for chip breaking (0.5 s). Confirm your entry.
Feed rate ?	
8 0 ENT	Enter the feed rate for drilling (F = 80 mm/min). Confirm your entry.
NC I	Drill.

Example: TAPPING

X coordinate of the hole:	30	mm
Y coordinate of the hole:	20	mm
Pitch p:	0.8	mm
Spindle speed S:	100	rpm
Clearance height:	+ 50	mm
Setup clearance Ⓐ :	3	mm
Workpiece surface:	0	mm
Thread depth Ⓑ :	– 20	mm
Dwell time:	0.4	s
Feed rate F = S • p:	80	mm/min



Preparation

➤ Pre-position the tool over the workpiece.

➤ For tapping **right-hand threads** activate the spindle with M 3.

Operating mode: POSITIONING WITH MDI

X	Select the X axis.
Nominal position value ?	
3 0 / Radius Comp.	Enter the nominal position value for pre-positioning in the X axis: X = + 30 mm and select tool radius compensation R 0.
NC I	Pre-position the tool in the X axis.
Y	Select the Y axis.
Nominal position value ?	
2 0	Enter the nominal position value for pre-positioning in the Y axis: Y = + 20 mm. Tool radius compensation is already set to R 0.
NC I	Pre-position the tool in the Y axis.

/	Go to the second soft-key row.
Tapping	Select Tapping.
Clearance height ?	
5 0 ENT	Enter the clearance height of the tool over the workpiece (+ 50 mm). Confirm your entry.
Setup clearance ?	
3 ENT	Enter setup clearance Ⓐ (3 mm). Confirm your entry.
Surface ?	
0 ENT	Enter the coordinate of the workpiece surface (0 mm). Confirm your entry.
Hole depth ?	
– 2 0 ENT	Enter hole depth Ⓑ (– 20 mm). Confirm your entry.
Dwell time ?	
0 . 4 ENT	Enter the dwell time (0.4 s). Confirm your entry.
Feed rate ?	
8 0 ENT	Enter the feed rate for tapping (80 mm/min). Confirm your entry.
NC I	Drill.

Hole patterns

The hole pattern functions **`Circle Pattern`** and **`Linear Pattern`** are available in the `POSITIONING WITH MDI` mode of operation.

Use the soft keys to select the desired hole pattern function and enter the required data. These data can usually be taken from the workpiece drawing (number of holes, coordinates of the first hole, etc.).

The TNC then calculates the positions of all holes in the pattern, and displays the pattern graphically on the screen.

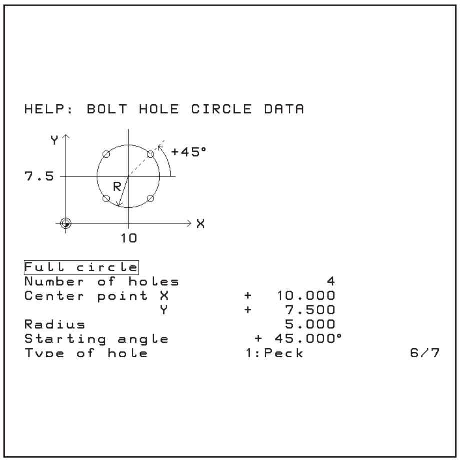


Fig. 4.3: On-screen operating instructions: graphic for bolt hole circle pattern (full circle)

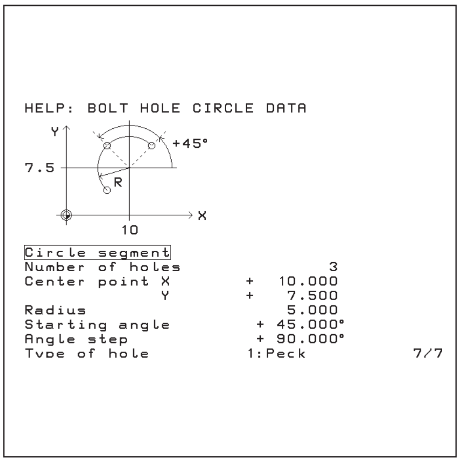


Fig. 4.4: On-screen operating instructions: graphic for bolt hole circle pattern (circle segment)

Type of hole

At the hole positions that were calculated for the pattern you can execute either

- pecking **or**
- tapping operations.

Enter the required data for pecking or tapping (see pages 43 to 47).

If you do not wish to drill at the calculated hole positions, or if you want to **drill the holes manually**:

➤ Choose the soft key `No Entry` for `Type of hole ?`.

Pre-positioning the drill

Pre-position the drill in the Z axis to a position above the workpiece surface. The TNC then pre-positions the drill in the X and Y axes (working plane) above each hole position.

Bolt hole circle patterns

If you are drilling a `Circle Pattern` in the `POSITIONING WITH MDI` mode of operation, enter the following data:

- Full circle or circle segment
- Number of holes
- Center point coordinates and radius of the circle
- Starting angle (position of first hole)
- Circle segment only: angle step between the holes
- Bore hole or tap hole

Linear hole patterns

If you are drilling a `Linear Pattern` in the `POSITIONING WITH MDI` mode of operation, enter the following data:

- Coordinates of the first hole
- Number of holes per row
- Spacing between holes on a row
- Angle between the first row and the X axis
- Number of rows
- Spacing between rows
- Bore hole or tap hole

Bolt hole circle patterns

Information required:
- Full circle or circle segment
- Number of holes
- Center point coordinates and radius of the circle
- Starting angle (position of first hole)
- Circle segment only: angle step between the holes
- Bore hole or tap hole

The TNC calculates the coordinates of all holes.

Bolt hole circle graphic

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

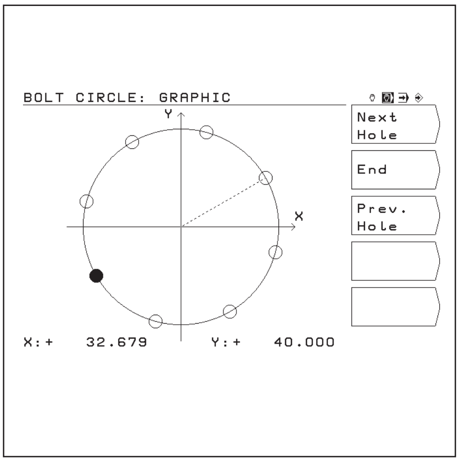
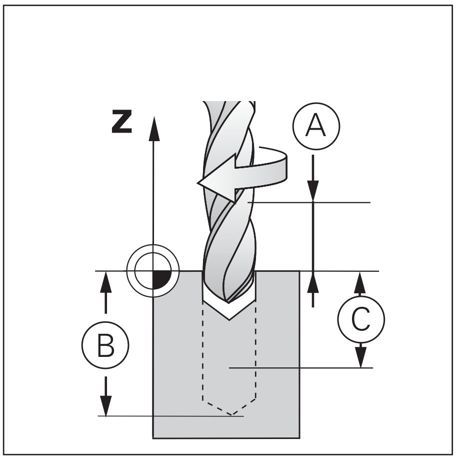


Fig. 4.5: TNC graphic for bolt hole circle patterns

Overview of functions

Function	Soft key/Key
Switch to full circle	Full Circle
Switch to circle segment	Circle Segment
Go to next-highest input line	↑
Go to next-lowest input line	↓
Confirm entry values	ENT

Example: Entering data and executing bolt hole circles



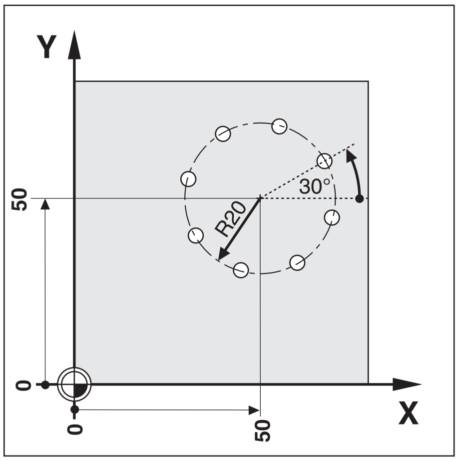
The work steps "Enter circle pattern data," "Display graphic" and "Drill" are described separately in this example.

Hole data
Enter the hole data separately (see pages 43 and 44) **before** entering the circle pattern data.

Clearance height:	+50	mm
Setup clearance Ⓐ :	3	mm
Workpiece surface:	0	mm
Hole depth Ⓑ :	– 20	mm
Pecking depth Ⓒ :	5	mm
Dwell time:	0.4	s
Feed rate:	80	mm/min

Circle pattern data

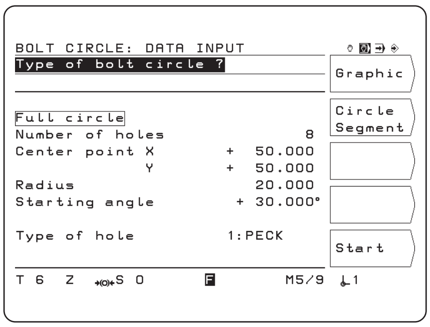
Number of holes:	8	
Center point coordinates:	X = 50	mm
	Y = 50	mm
Bolt hole circle radius:	20	mm
Starting angle: angle between X axis and first hole	30°	



1st step: Enter circle pattern data

Operating mode: `POSITIONING WITH MDI`

⎘ / ⎘	Go to the second soft-key row in the operating mode `POSITIONING WITH MDI`.
`Circle Pattern`	Select `Circle Pattern`.
`Full Circle`	Select `Full Circle`.

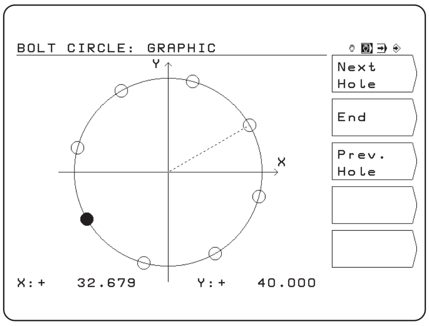


↓	Enter the data and call the dialog.
Number of holes ?	
8 ENT	Enter the number of holes (8). Confirm your entry.
Center point X ?	
5 0 ENT	Enter the X coordinate of the center of the bolt hole circle (X = 50 mm). Confirm your entry.
Center point Y ?	
5 0 ENT	Enter the Y coordinate of the center of the bolt hole circle (Y = 50 mm). Confirm your entry.
Radius ?	
2 0 ENT	Enter the radius of the bolt hole circle (20 mm). Confirm you entry.
Starting angle ?	
3 0 ENT	Enter the starting angle from the X axis to the first hole (30°). Confirm your entry.
Type of hole ?	
Pecking	Choose `Pecking` for drilling bore holes at the hole positions in the pattern.

2nd step: Display graphic

The graphic makes it easy to verify the entered data.
The solid circle represents the currently selected hole.

Graphic	The TNC displays the bolt hole circle graphically on the screen. Here, a full circle with 8 holes is shown. The first hole is at 30°. The coordinates of the hole are given at the bottom of the screen.



> The **direction of rotation** for bolt hole circle graphics is influenced with a user parameter (see Chapter 13).
> The TNC can **mirror** the coordinate axes for bolt hole circle graphics (see Chapter 13).

3rd step: Drill

Before you start drilling verify the data entered in the drilling cycle!

> The **direction of rotation** for bolt hole circles is influenced with a user parameter (see Chapter 13).

Start	Start the bolt hole circle function.
NC I	Pre-position in the first coordinate axis.
NC I	Pre-position in the second coordinate axis.
NC I	Drill. The TNC drills the bolt hole as defined by the input data for `Pecking` (or `Tapping`).
NC I	Drill the next hole and all remaining holes.

Functions for drilling and graphic

Function	Soft key
Go to next hole	Next Hole
Return to last hole	Last Hole
End graphic/drilling	End

Linear hole patterns

Information required:
- Coordinates of the first hole
- Number of holes per row
- Spacing between holes on a row
- Angle between the first row and the angle reference axis
- Number of rows
- Spacing between rows
- Bore hole or tap hole

The TNC calculates the coordinates of all holes.

Linear hole pattern graphic

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

Overview of functions

Function	Key
Go to the next-highest input line	↑
Go to the next-lowest input line	↓
Confirm entry values	ENT

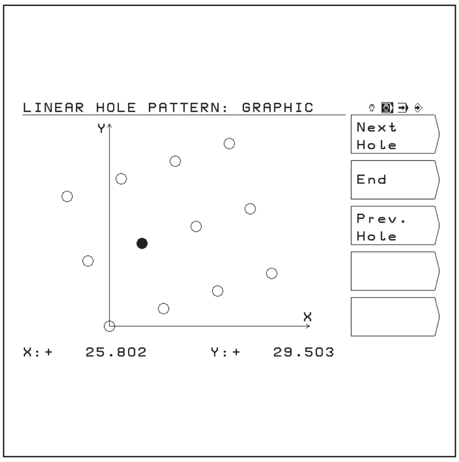
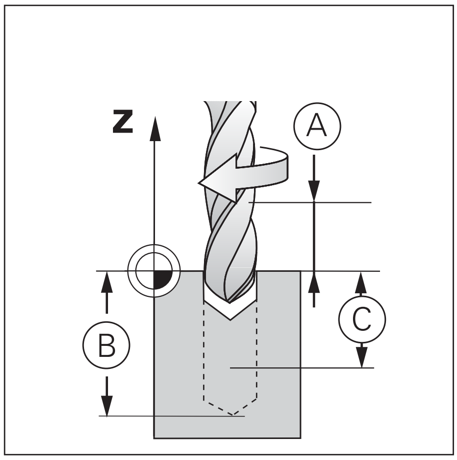


Fig. 4.6: TNC graphic for linear hole patterns

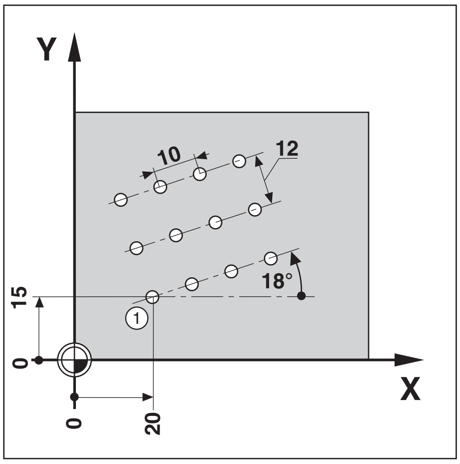
Example: Entering data and executing rows of holes



The work steps "Enter linear pattern data," "Display graphic" and "Drill" are described separately in this example.

Hole data
Enter the hole data separately (see pages 43 and 44) **before** entering the linear pattern data.

Clearance height:	+50 mm
Setup clearance (A) :	3 mm
Workpiece surface:	0 mm
Hole depth (B) :	– 20 mm
Pecking depth (C) :	5 mm
Dwell time:	0.4 s
Feed rate:	80 mm/min



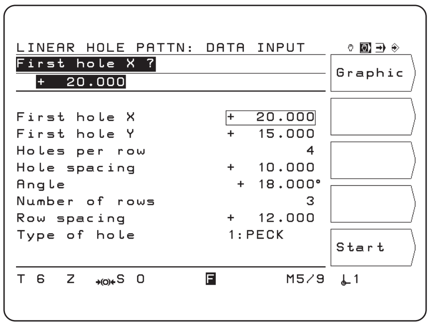
Linear pattern data

X coordinate of hole (1) :	X = 20 mm
Y coordinate of hole (1) :	Y = 15 mm
Number of holes per row:	4
Hole spacing:	+10 mm
Angle between rows and X axis:	18°
Number or rows:	3
Row spacing:	+12 mm

1st step: Enter linear pattern data

Operating mode: `POSITIONING WITH MDI`

/	Go to the second soft-key row in the operating mode `POSITIONING WITH MDI`.
Linear Pattern	Select `Linear Pattern`.

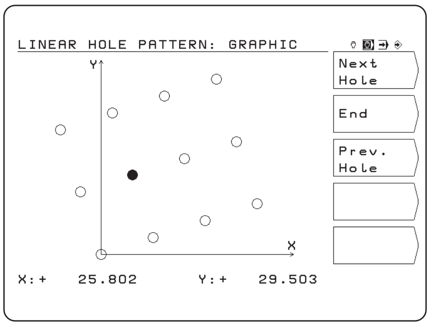


	`1st hole X ?`
2 0 ENT	Enter the X coordinate of hole ① (X = 20 mm). Confirm your entry.
	`1st hole Y ?`
1 5 ENT	Enter the Y coordinate of hole ① (Y = 15 mm). Confirm your entry.
	`Holes per row ?`
4 ENT	Enter the number of holes per row (4). Confirm your entry.
	`Hole spacing ?`
1 0 ENT	Enter the spacing between holes in the row (10 mm). Confirm your entry.
	`Angle ?`
1 8 ENT	Enter the angle between the X axis and the hole pattern (18°). Confirm your entry.
	`Number of rows ?`
3 ENT	Enter the number of rows (3). Confirm your entry.
	`Row spacing ?`
1 2 ENT	Enter the spacing between rows (12 mm). Confirm your entry.
	`Type of hole ?`
`Pecking`	Choose `Pecking` for drilling bore holes at the hole positions in the pattern.

2nd step: Display graphic

The graphic makes it easy to verify the entered data.
The solid circle represents the currently selected hole.

Graphic	The TNC displays the pattern graphically on the screen. Here, 3 rows of 4 holes are shown. 1st hole at X=20 mm, Y=10 mm Hole spacing 10 mm Angle between rows and X axis: 18° Row spacing 12 mm Coordinates of the current hole are shown at the bottom of the screen.



The TNC can **mirror** the coordinate axes for linear hole pattern graphics if the corresponding user parameter is set (see Chapter 13).

3rd step: Drill

Before you start drilling verify the data entered in the drilling cycle!

Start	Start the linear hole pattern function.
NC I	Pre-position in the first coordinate axis.
NC I	Pre-position in the second coordinate axis.
NC I	Drill. The TNC drills the bolt hole as defined by the input data for `Pecking` (or `Tapping`).
NC I	Drill the next and all remaining holes.

Functions for drilling and graphic

Function	Soft key
Go to next hole	Next Hole
Return to last hole	Last Hole
End graphic/drilling	End

Rectangular pocket milling

The TNC cycle for rectangular pocket milling is available in the POSITIONING WITH MDI mode of operation.

The input data for milling a rectangular pocket can also be entered as a "cycle" in a part program (see Chapter 7).

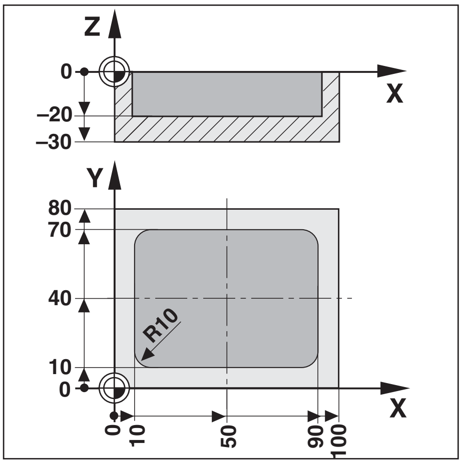
Select the "Pocket Milling" soft key on the second soft-key row and enter the required data. These data can usually be taken from the workpiece drawing (side length, depth of the pocket, etc.).

The TNC controls the machine tool and calculates the tool path for area clearance.

For the procedure and input data required for programming a rectangular pocket, see Chapter 7.

Example: RECTANGULAR POCKET

Clearance height:	+ 80	mm
Safety clearance:	2	mm
Workpiece surface:	+ 0	mm
Milling depth:	– 20	mm
Pecking depth:	7	mm
Pecking feed rate:	80	mm/min
Pocket center in X:	50	mm
Pocket center in Y:	40	mm
Side length in X:	80	mm
Side length in Y:	60	mm
Milling feed rate:	100	mm/min
Direction of milling rotation:	0: CLIMB	
Finishing allowance:	0.5	mm



Operating mode: POSITIONING WITH MDI

/	Go to the second soft-key row.
Pocket Milling	Select Pocket Milling.
Clearance height ?	
8 0 ENT	Enter the clearance height of the tool over the workpiece (HEIGHT +80 mm). Confirm your entry.
Setup clearance ?	
2 ENT	Enter the setup clearance (DIST 2 mm). Confirm your entry.
Surface ?	
0 ENT	Enter the coordinate of the workpiece surface (SURF 0 mm). Confirm your entry.
. . .	
NC I	After you have entered all the data, start the rectangular pocket milling cycle.

5
Programming

Operating mode PROGRAMMING AND EDITING

In the `PROGRAMMING AND EDITING` mode of operation you can store the individual work steps that are required for recurring machining operations, for example for small-lot production.

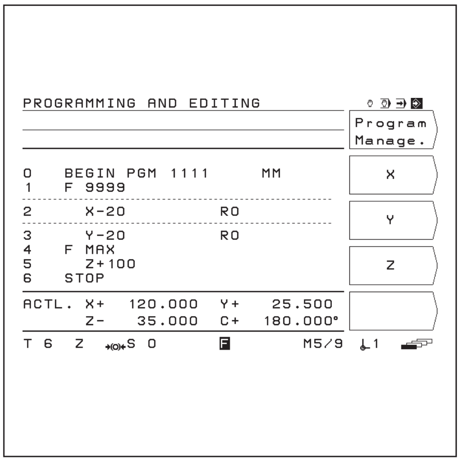


Fig. 5.1: The first soft-key row in the operating mode `PROGRAMMING AND EDITING`

Programs in the TNC

Programs contain the work steps for workpiece machining. You can edit programs, add work steps and run them as often as you wish.

The `External` mode enables you to store programs with the HEIDENHAIN FE 401 floppy disk unit and load them into the TNC again on demand—you don't need to retype them. You can also transfer programs to a personal computer or printer.

Storage capacity
The TNC 124 can store a maximum of 20 programs with a total of 2000 NC blocks. A single program can contain a maximum of 1000 NC blocks.

Position display during programming

In the `PROGRAMMING AND EDITING` mode of operation, the TNC continuously displays the current positions at the bottom of the screen to left of the lowest soft key.

Programmable functions

- Nominal position values
- Feed rate F, spindle speed S and miscellaneous function M
- Tool call
- Pecking and tapping cycles
- Bolt hole circle and linear hole patterns
- Program section repeats:
  A section of a program only has to be entered once and can then be run up to 999 times in succession.
- Subprograms:
  A section of a program only has to be entered once and can then be run at various points in the program.
- Datum call
- Dwell time
- Interrupt program

Transfer position: Teach-In mode

This mode allows you to transfer the actual positions of the tool directly into a program, such as the nominal positions for workpiece machining, etc.
In many cases the Teach-In function will save you considerable programming work.

What happens with finished programs?

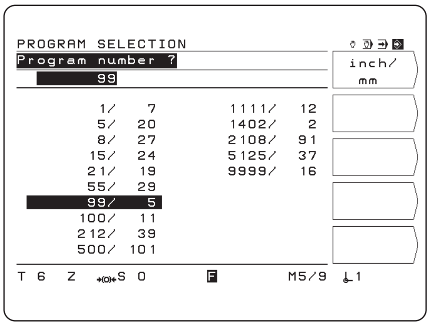
For workpiece machining, programs are executed in the operating mode `PROGRAM RUN`. See Chapter 10 for an explanation of this mode.

Entering a program number

Select a program and identify it by a number between 0 and 9999 9999 which you assign it.

Operating mode: PROGRAMMING AND EDITING

Program Manage.	Select Program Manage.
Program Number	Go to the program directory.



Program number ?

1	Create a new program or select an existing program, such as program number 1.
or	or
→ / ↑ / ↓	Select an existing program with the cursor keys.
inch/ mm	Choose the unit of measurement.
ENT	Confirm your entry. The selected program can now be entered, edited and run.

> When you select the unit of measurement with the soft key inch/mm, the TNC overwrites the user parameter inch/mm.

Program directory

The program directory appears when you choose the soft key Program Number. The number in front of the slash is the program number, the number behind the slash is the number of blocks in the program.

A program always contains at least two blocks.

Deleting programs

If you no longer wish to keep a program in memory, you can **delete** it:

- ➤ Press the soft key Program Manage.
- ➤ Press the soft key Delete Program.
- ➤ Enter the program number.
- ➤ Press ENT to delete the program.

Editing programs

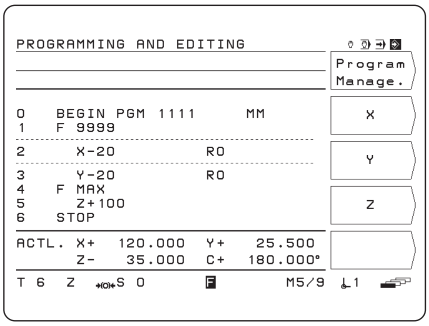
Operating mode: `PROGRAMMING AND EDITING`

Program Manage.	Select a program (see previous page).

/

The first soft-key row provides functions for

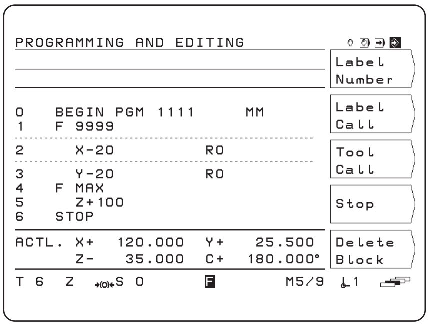
- Selecting program management
- Entering coordinates



/

The second soft-key row provides the following functions:

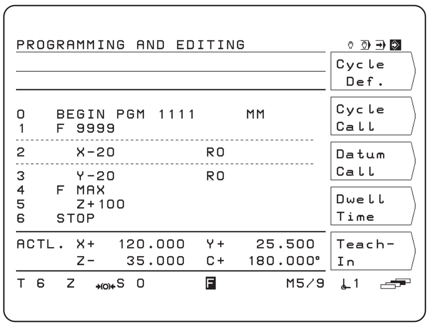
- Enter labels for subprograms and program section repeats
- Call tool data
- Interrupt program with `Stop`
- Delete program blocks



/

The third soft-key row provides cycles for entering:

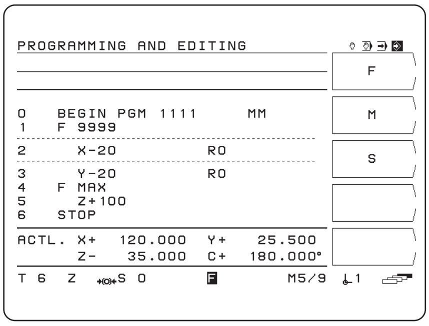
- Cycle definition for pecking, tapping, bolt hole circles and linear hole patterns
- Cycle call
- Datum call
- Dwell time
- Teach-In



/

The fourth soft-key row provides the functions

- Feed rate F
- Miscellaneous function M
- Spindle speed S



Editing program blocks

Current block

The current block is shown between the two dashed lines. New blocks are inserted behind the current block. When the `END PGM` block is between the dashed lines, no new blocks can be inserted.

Overview of functions

Function	Soft key/Key
Go up one block	↑
Go down one block	↓
Clear numerical entry	CE
Delete current block	Delete Block

Going directly to a program block

Scrolling to the desired block with the arrow keys can be time-consuming with long programs. A quicker way is to use the GOTO function. This enables you to move directly to the block you wish to change or add new blocks behind.

Operating mode: `PROGRAMMING AND EDITING`

GOTO	Press the GOTO key.
Block number ?	
5 8	Enter a block number, such as 58.
ENT	Confirm your entry. Block number 58 is now the currently selected block.

Editing existing programs

You can edit existing programs, for example to correct keying errors. The TNC supports you with plain language dialogs—just as when you are creating a new program.

Confirm your changes

You **must** confirm each change with the ENT key for it to become effective!

Example: Changing a program number

- ➤ Select the `BEGIN` or `END` block.
- ➤ Enter a new program number.
- ➤ Confirm the change with ENT.

Example: Editing a program block

Operating mode: `PROGRAMMING AND EDITING`

↑ / ↓	Move to the block you wish to change.
→	Select the block.
2 0	Edit the block, for example enter a new nominal position value (here, 20).
ENT	Confirm the change.

Overview of functions

Function	Key
Select the next-lowest program block	↓
Select the next-highest program block	↑
Go directly to program block number	GOTO □
Select program block to edit	→
Confirm change	ENT

Deleting program blocks

You can delete any blocks in existing programs except the `BEGIN` and `END` blocks.

When a block is deleted, the TNC automatically renumbers the remaining blocks. The block before the deleted block then becomes the current block.

Example: Deleting a program block

Operating mode: `PROGRAMMING AND EDITING`

↑ / ↓	Move to the block you wish to delete (or use the GOTO key).
/	Go to the second soft-key row.
Delete Block	Press `Delete Block`.

It is also possible to delete an entire **program section:**

➤ Select the last block of the program section to be deleted.

➤ Press the soft key `Delete Block` repeatedly until all blocks in the program section have been deleted.

Feed rate F, spindle speed S and miscellaneous function M

Besides the geometry for workpiece machining, you must also enter the following information:

- Feed rate F in [mm/min]
- Miscellaneous function M
- Spindle speed S in [rpm]

The feed rate F, miscellaneous function M and spindle speed S are programmed in separate blocks and become effective as soon as the TNC has executed the blocks in which they are programmed.
These program blocks must be entered in the program **before** the positioning blocks for which they are intended.

Entering the feed rate F

The feed rate F is "modally" effective. This means that the entered feed rate remains in effect until a new feed rate is programmed.

Exception: Rapid traverse F MAX

Rapid traverse F MAX
You can also move the machine axes at rapid traverse (F MAX). The feed rate for rapid traverse F MAX is preset in a machine parameter by the machine manufacturer.
F MAX is **not** modally effective.
After the block with F MAX is executed, the feed rate returns to the value that was programmed previously.

Programming example:

Operating mode: `PROGRAMMING AND EDITING`

/	Go to the fourth soft-key row.
F	Select `Feed rate F`.
Feed rate ?	
5 0 0 ENT	Enter the feed rate F, such as F = 500 mm/min. Confirm entry. Input range: 0 to 30 000 mm/min.
or	or
F MAX	Select rapid traverse `F MAX`.

The feed rate can be varied infinitely during program run by turning the knob for feed rate override on the TNC control panel.

Entering the spindle speed S

The machine manufacturer determines which spindle speeds are allowed on your TNC.

The spindle speed S is "modally" effective. This means that the entered spindle speed remains in effect until a new spindle speed is programmed.

Programming example

Operating mode: `PROGRAMMING AND EDITING`

/	Go to the fourth soft-key row.
S	Select `Spindle speed S`.
Spindle speed ?	
9 9 0 ENT	Enter the spindle speed S, such as S = 990 rpm. Confirm entry. Input range: 0 to 9999.999 rpm.

The spindle speed can be varied infinitely during program run by turning the knob for spindle speed override on the TNC control panel.

Entering a miscellaneous function M

With the miscellaneous functions (M functions) you can influence, for example, direction of spindle rotation and program run.
Chapter 14 provides an overview of all miscellaneous functions that can be programmed on the TNC 124.

The machine manufacturer determines which miscellaneous functions are available on your TNC and which functions they have.

Programming example

Operating mode: `PROGRAMMING AND EDITING`

/	Go to the fourth soft-key row.
M	Select `Miscellaneous function M`.
Miscellaneous function M ?	
3 ENT	Select the miscellaneous function, such as M 3 (spindle ON, clockwise). Confirm entry.

Entering program interruptions

You can divide a program into sections with stop marks. The TNC then only executes the next block when you resume program run.

Operating mode: PROGRAMMING AND EDITING

/	Go to the second soft-key row.
Stop	Press STOP to insert a program interruption.

Resuming program run after an interruption

➤ Press the NC-I key.

Calling the tool data in a program

Chapter 3 explained how to enter the length and radius of your tools in the tool table.

The tool data stored in the table can also be called from a program. Then if you change the tool during program run you don't need to select the new tool data from the tool table every time.

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

You define the tool axis for program run in the program.

> If you enter a different tool axis in the program than is stored in the table, the TNC stores the new tool axis in the table.

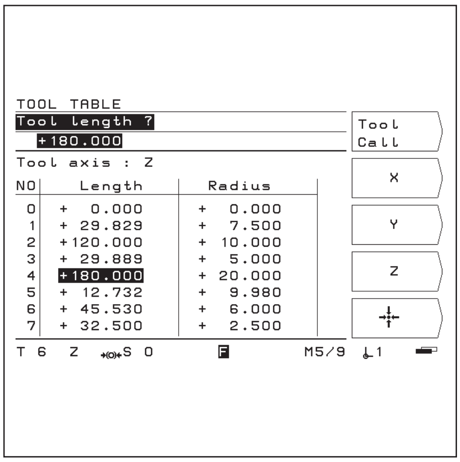


Fig. 5.2: The tool table on the TNC screen

Operating mode: `PROGRAMMING AND EDITING`

/	Go to the second soft-key row.
Tool Call	Call tool data from the tool table.

Tool number ?

4 ENT	Enter the tool number (such as 4) under which the tool data are stored in the tool table. Confirm entry. Input range: 0 to 99.

Tool axis ?

Z	Enter the tool axis (such as Z). The program contains the tool call block `TOOL CALL 4 Z`.
or	or
No Entry	Choose `No Entry` for the `Tool axis`, if the program already contains a `TOOL CALL` block with tool data. The program contains the tool call block `TOOL CALL 4`.

Working without TOOL CALL

If a part program is written without `TOOL CALL` the TNC will use the data of the tool that was programmed previously.
When you are changing tools, you can also go to the tool table from the operating mode `PROGRAM RUN` to call the new tool data.

Calling datum points

The TNC 124 can store up to 99 datum points in a datum table. You can call a datum point from the datum table during program run by simply pressing the soft key `Datum Call` and entering the block `DATUM XX`. This automatically calls the datum point entered for XX during program run.

Operating mode: `PROGRAMMING AND EDITING`

/	Go to the third soft-key row.
Datum Call	Call a datum point from the table.
Datum number?	
5 ENT	Enter the datum number (such as 5). Confirm entry. Input range: 1 to 99.

Entering dwell time

You can enter a dwell time in the part program by pressing the soft key `Dwell Time` and defining the block `DWELL XXXX.XXX`. When the `DWELL` block is executed, continuation of the running program is delayed by the time entered in seconds for `DWELL`.

Operating mode: `PROGRAMMING AND EDITING`

/	Go to the third soft-key row.
Dwell Time	Call dwell time.
Dwell time in seconds?	
8 ENT	Enter the dwell time in seconds (such as 8). Confirm entry. Input range: 0 to 9999.999.

6
Programming Workpiece Positions

Entering workpiece positions

For many simple machining processes it is often sufficient to simply describe the workpiece to be machined by the coordinates of the positions to which the tool should move.

There are two possibilities of entering these coordinates in a program:
- Keying in the coordinates with the keyboard, or
- Transferring the tool position with the `Teach-In` function

Entries for a complete part program

Having the TNC execute a machining process requires more than entering coordinates in a program. A complete part program requires the following data:
- A `BEGIN` block and an `END` block (automatically generated by the TNC)
- Feed rate F
- Miscellaneous function M
- Spindle speed S
- Calling the tool with `TOOL CALL`

Entering feed rate F, miscellaneous function M, spindle speed S and `TOOL CALL` in a part program is described in Chapter 5.

Important information on programming and machining

The following information is intended to help you in quickly and easily machining the programmed workpiece.

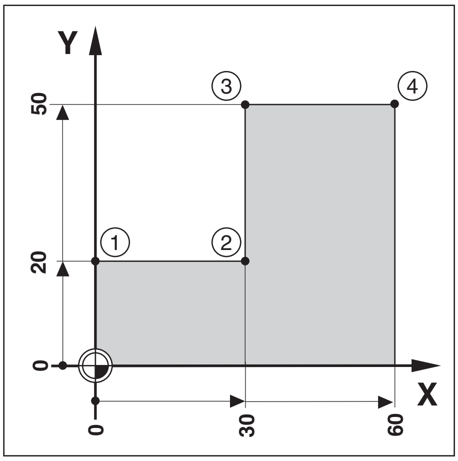
Movements of tool and workpiece
During workpiece machining on a milling or drilling machine, an axis position is changed either by moving the tool or by moving the machine table on which the workpiece is fixed.

> When entering tool movements in a part program you always program as if the tool is moving and the workpiece is stationary.

Pre-positioning
Pre-position the tool to prevent the possibility of damaging the tool or workpiece. The best pre-position lies on the extension of the tool path.

Feed rate F and spindle speed S
Adjust the feed rate F and spindle speed S to your tool, workpiece material and machining operation.
The TNC then calculates the feed rate F and spindle speed S with the **INFO** function (see Chapter 12).
In the appendix you will find a diagram which will aid you in selecting the appropriate feed rate F for tapping.

Programming example: Milling a shoulder



The coordinates are programmed in absolute dimensions.
The datum is the workpiece zero.

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

Summary of all programming steps

➤ In the main menu PROGRAMMING AND EDITING go to Program Manage.
➤ Key in the number of the program you want to work on, and press ENT.
➤ Enter the nominal positions.

Running a finished program

When a program is finished it can be run in the PROGRAM RUN mode (see Chapter 10) .

Example of entry: Entering a nominal position into a program (block 11 in this example)

X	Select the coordinate axis (X axis).

Nominal position value ?

3 0 Radius Comp.	Enter the nominal position value, for example 30 mm and select tool radius compensation R –.
ENT	Confirm the entry. The nominal position is now the current block (between the dashed lines).

Program blocks

0	BEGIN PGM 10	MM	Start of program, program number and unit of measurement
1	F 9999		High feed rate for pre-positioning
2	Z+20		Clearance height
3	X–20	R0	Pre-position the tool in the X axis
4	Y–20	R0	Pre-position the tool in the Y axis
5	Z–10		Move tool to milling depth
6	TOOL CALL 1 Z		Call the tool, such as tool 1, tool axis Z
7	S 1000		Spindle speed
8	M 3		Spindle ON, clockwise
9	F 200		Machining feed rate
10	Y+20	R+	Y coordinate, corner 1
11	X+30	R–	X coordinate, corner 2
12	Y+50	R+	Y coordinate, corner 3
13	X+60	R+	X coordinate, corner 4
14	F 9999		High feed rate for retracting
15	Z+20		Clearance height
16	M 2		Stop program run, spindle OFF, coolant OFF
17	END PGM 10	MM	End of program, program number and unit of measurement

Transferring positions: Teach-In mode

Teach-In programming offers the following two options:

- Enter nominal position, transfer nominal position to program, move to position.
- Move to a position and transfer the actual value to a program via soft key or through the actual-value-capture key on the handwheel.

You can change transferred position values during Teach-In.

Preparation

➤ With `Program number` select the program you want transfer positions to.

➤ Select the tool data from the tool table.

Feed rate F for Teach-In

Before starting the Teach-In process define the feed rate at which the tool should move during Teach-In:

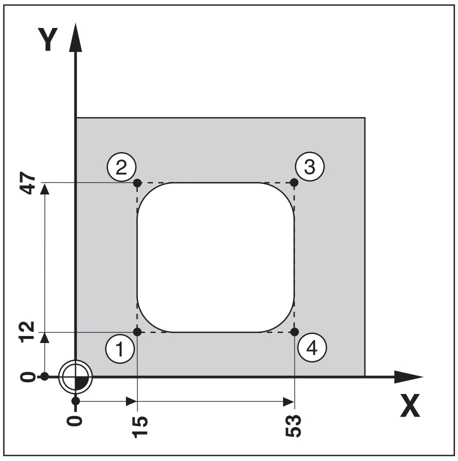
➤ Select the Teach-In function and enter a block with the desired feed rate F first.

➤ Press the NC-I key.

Overview of functions

Function	Soft key/Key
Go to the next block	↓
Go to the previous block	↑
Delete the current block	Delete Block

Programming example: Generate a program while machining a pocket



With Teach-In you first machine a workpiece according to the workpiece drawing dimensions.

The TNC then transfers the coordinates directly into the program. Pre-positioning and retraction movements can be selected as desired and entered like drawing dimensions.

Corner point 1 :	X = 15 mm	Y = 12 mm
Corner point 2 :	X = 15 mm	Y = 47 mm
Corner point 3 :	X = 53 mm	Y = 47 mm
Corner point 4 :	X = 53 mm	Y = 12 mm

Pocket depth: Z = – 10 mm (for example)

Operating mode: `PROGRAMMING AND EDITING`

`Teach-In`	Select `Teach-In`.

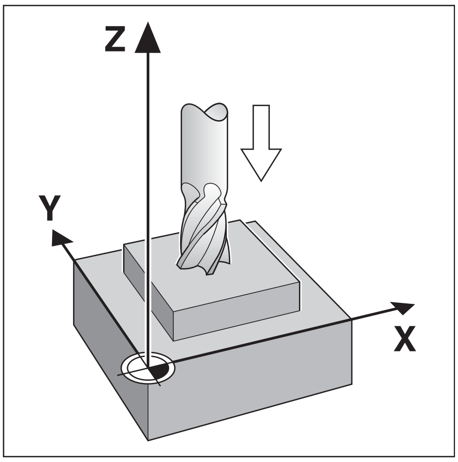
Example: Transferring the Y coordinate of corner point 3 into a program

`Y`	Select the coordinate axis (Y axis).

`Nominal position value ?`

4 7 `Radius Comp.`	Enter the nominal position value (such as 47 mm) and select tool radius compensation R –.
NC I	Move to the programmed coordinate. Then enter and transfer any other coordinates.

Programming example: Touch island with tool and transfer positions to program



This example illustrates how to generate a program containing the actual positions of the tool.

When you then **run** the program:

➤ Use a tool which has the same radius as the tool you used during the Teach-In process.

➤ If you use a different tool, you must enter all program blocks with radius compensation. Then enter the difference between the radii of the two tools as the tool radius for machining:

	Radius of the tool for machining
–	Radius of the tool for Teach-In
=	Tool radius to be entered for machining

Selecting radius compensation

The current radius compensation is highlighted at the top of the screen. If you wish to change the radius compensation:

➤ Press the soft key `Radius Comp`.

Operating mode: `PROGRAMMING AND EDITING`

Teach-In	Select `Teach-In`.
/	Page to the second soft-key row.

Example: Transfer Z coordinate (workpiece surface) to a program

Z′	Move the tool until it touches the workpiece surface.
Capture Z	Store the position in the tool axis (Z) with the soft key at the TNC
or	or
Z	with the actual-position-capture key on the handwheel.

Changing nominal positions after they have been transferred

Positions which you have transferred into a program with Teach-In can be changed. It is not necessary to leave the Teach-In mode to do so.

Enter the new value in the input line.

Example: Changing a block transferred with Teach-In

Operating mode: `PROGRAMMING AND EDITING, Teach-In`

↑ / ↓	With the arrow keys (or GOTO), move to the block you wish to change.
→	Select the block.

Nominal position value ?

3 0 Radius Comp.	Enter a new nominal position value and change the tool radius compensation (for example).
ENT	Confirm your changes.

Functions for changing a Teach-In program

Function	Soft key
Enter feed rate F	F
Enter miscellaneous function M	M
Enter spindle speed S	S
Delete current block	Delete Block

7
Drilling, Milling Cycles and Hole Patterns in Programs

The cycles for pecking or tapping, hole patterns, and rectangular pocket milling can also be written to a program (see Chapter 4). Each piece of information is then stored in a separate program block. These blocks are identified by `CYCL` after the block number, followed by a number.

The cycles contain all information required by the TNC for machining a hole, hole pattern or rectangular pocket.

The TNC 124 features six different cycles:

Drilling cycles
- `CYCL 1.0 PECKING`
- `CYCL 2.0 TAPPING`

Hole patterns
- `CYCL 5.0 FULL CIRCLE`
- `CYCL 6.0 CIRCLE SEGMENT`
- `CYCL 7.0 LINEAR HOLE PATTN`

Rectangular pocket milling
- `CYCL 4.0 RECTANGULAR POCKET`

Cycles must be complete

Do not delete any blocks from a cycle because this will result in the error message `CYCLE INCOMPLETE` when the program is executed.

Drilling cycles must be called

The TNC runs a **drilling cycle** whenever it reaches a cycle call (`CYCL CALL`) during execution of the program. A cycle call always calls the drilling cycle that was programmed before the cycle call.

The TNC automatically executes a **hole pattern** or **rectangular pocket** as soon as it reaches it during execution of the program. If you wish to repeatedly execute hole patterns or rectangular pockets, you must enter the data repeatedly or write them in a subprogram (see Chapter 8).

Entering cycles

Press the `Cycle Def.` soft key in the third soft-key row and select the desired cycle. The TNC automatically asks for all data required for executing the cycle.

Entering a cycle call

A drilling cycle must be called at the location in a part program at which the cycle is to be executed.

Operating mode: `PROGRAMMING AND EDITING`

/	Go to the third soft-key row.
Cycle Call	Enter a cycle call (`CYCL CALL`).

Drilling cycles in programs

The following two cycles are available on the TNC 124:

- `CYCL 1.0 PECKING`
- `CYCL 2.0 TAPPING`

Cycle `1.0 PECKING`

Cycle `1.0 PECKING` is used for drilling holes in several infeeds.

During machining the TNC advances the tool in several infeeds, retracting the tool each time to setup clearance.

Cycle `2.0 TAPPING`

The `TAPPING` cycle requires a **floating tap holder**.

Cycle `2.0 TAPPING` is used for cutting threads.

The thread is cut in one pass. After a dwell time at the end of thread, the direction of spindle rotation is reversed and the tool retracted.

Signs for the input values in the drilling cycles

Enter the "clearance height" H and the coordinate of the workpiece surface O as absolute values—**together with the algebraic sign**.

The **algebraic sign for hole depth** (thread length) B determines the working direction. If you are drilling in the negative axis direction, enter a negative sign for hole depth.

Fig. 7.1 also illustrates setup clearance A and the infeed depth C .

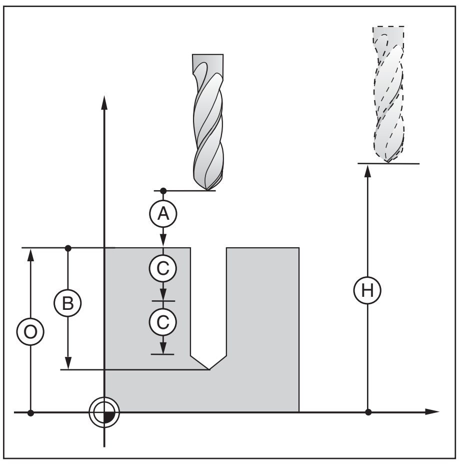


Fig. 7.1: Absolute and incremental input values for drilling cycles

Pre-positioning the drill

Before executing the cycle, pre-position the drill in the tool axis and in the working plane. The coordinates for pre-positioning can be entered into the program before the cycle.

PECKING

If you program Cycle `1.0 PECKING`, the TNC drills to the programmed hole depth in several infeeds.

Process

The pecking cycle is illustrated in Fig. 7.2 and Fig. 7.3.

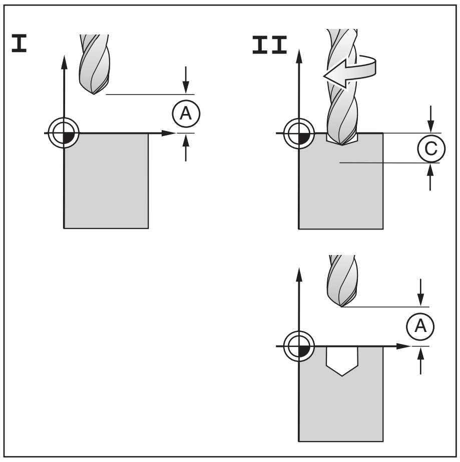


Fig. 7.2: Steps `I` and `II` in Cycle `1.0 PECKING`

I:
The TNC pre-positions the tool at setup clearance Ⓐ above the workpiece surface.

II:
The tool drills to the first pecking depth © at the programmed machining feed rate F. After reaching the first pecking depth, the tool retracts at rapid traverse (F MAX) to setup clearance Ⓐ .

III:
The TNC pre-positions the tool at rapid traverse to the first infeed depth © , minus the advanced stop distance ⓣ. The tool then advances with another infeed © .

IV:
The TNC retracts the tool again and repeats the drilling process (drilling/retracting) until the programmed hole depth Ⓑ is reached.

After a dwell time at the hole bottom, the tool is retracted to clearance height at rapid traverse (F MAX) for chip breaking.

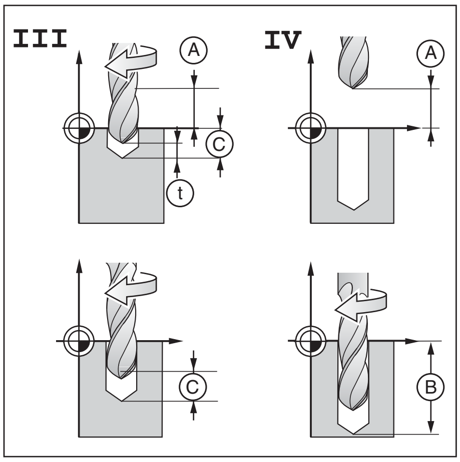


Fig. 7.3: Steps `III` and `IV` in Cycle `1.0 PECKING`

Advanced stop distance ⓣ

The advanced stop distance ⓣ for the drilling operation is automatically calculated by the TNC:

Hole depth up to 30 mm:	ⓣ = 0.6 mm
Hole depth between 30 mm and 350 mm:	ⓣ = 0.02 • hole depth
Hole depth exceeding 350 mm:	ⓣ = 7 mm

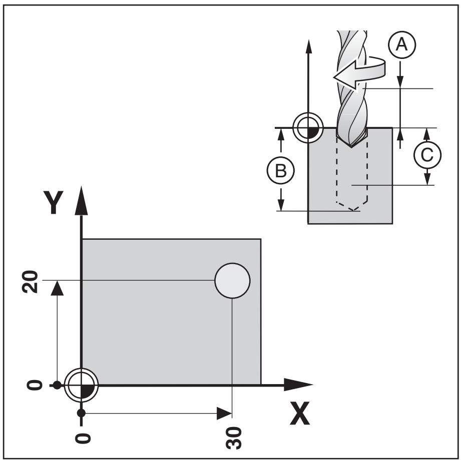
Input data for Cycle `1.0 PECKING`

- Clearance height - `HEIGHT`
  Position in the tool axis at which the TNC can move the tool in the working plane without damaging the workpiece.
- Setup clearance - `DIST` Ⓐ
  The TNC advances the tool from clearance height to setup clearance at rapid traverse.
- Workpiece surface - `SURF`
  Absolute coordinate of the workpiece surface.
- Hole depth - `DEPTH` Ⓑ
  Distance between workpiece surface and bottom of hole (tip of drill taper).
- Pecking depth - `PECKG` ©
  Infeed per cut.
- Dwell time - `DWELL` in [s]
  Amount of time the tool remains at the hole depth for cutting free the drill taper.
- Feed rate - `F` in [mm/min]
  Traversing speed of the tool while drilling.

Hole depth and infeed depth

The infeed depth does not have to be a multiple of the hole depth. If the infeed depth is programmed greater than the hole depth, or equals the hole depth, the tool will drill to the programmed hole depth in one operation.

Programming example: Cycle `1.0 PECKING`



X coordinate of the hole:	30 mm
Y coordinate of the hole:	20 mm
Hole diameter:	6 mm
Clearance height `HEIGHT`:	+ 50 mm
Setup clearance `DIST` Ⓐ :	2 mm
Coordinate of the workpiece surface `SURF`:	0 mm
Hole depth `DEPTH` Ⓑ:	– 15 mm
Pecking depth `PECKG` Ⓒ:	5 mm
Dwell time `DWELL`:	0.5 s
Machining feed rate `F`:	80 mm/min

Example: Entering Cycle 1.0 `PECKING` in a part program

Operating mode: `PROGRAMMING AND EDITING`

[page key] / [page key]	Page to the third soft-key row.
Cycle Def.	Select `Cycle Definition`.
Pecking	Enter Cycle `1.0 PECKING` in a part program.

`Clearance height ?`

5 0 ENT	Enter the clearance height (`HEIGHT = 50` mm). Confirm your entry.

`Setup clearance ?`

2 ENT	Enter the setup clearance Ⓐ (`DIST = 2` mm). Confirm your entry.

`Workpiece surface ?`

0 ENT	Enter the coordinate of the workpiece surface (`SURF = 0` mm). Confirm your entry.

`Hole depth ?`

– 1 5 ENT	Enter the hole depth Ⓑ (`DEPTH = –15` mm). Confirm your entry.

`Pecking depth ?`

5 ENT	Enter the pecking depth Ⓒ (`PECKG = 5` mm). Confirm your entry.

Dwell time ?	
0 . 5 ENT	Enter the dwell time for chip breaking (DWELL = 0.5 s). Confirm your entry.

Feed rate ?	
8 0 ENT	Enter the feed rate for drilling (F = 80 mm/min). Confirm your entry.

Program blocks

0	BEGIN PGM 20 MM	Start of program, program number, unit of measurement
1	F 9999	High feed rate for pre-positioning
2	Z+600	Tool-change position
3	X+30	Pre-positioning in the X axis
4	Y+20	Pre-positioning in the Y axis
5	TOOL CALL 8 Z	Call the tool for pecking, such as tool 8, tool axis Z
6	S 1500	Spindle speed
7	M 3	Spindle ON, clockwise
8	**CYCL 1.0 PECKING**	Cycle data for Cycle 1.0 PECKING follow
9	**CYCL 1.1 HEIGHT +50**	Clearance height
10	**CYCL 1.2 DIST 2**	Setup clearance above the workpiece surface
11	**CYCL 1.3 SURF + 0**	Absolute coordinate of the workpiece surface
12	**CYCL 1.4 DEPTH -15**	Hole depth
13	**CYCL 1.5 PECKG 5**	Depth per infeed
14	**CYCL 1.6 DWELL 0.5**	Dwell time at bottom of hole
15	**CYCL 1.7 F 80**	Machining feed rate
16	CYCL CALL	Cycle call
17	M 2	Stop program run, spindle STOP, coolant OFF
18	END PGM 20 MM	End of program, program number, unit of measurement

Cycle 1.0 PECKING is executed in the operating mode PROGRAM RUN (see Chapter 10).

TAPPING

With Cycle 2.0 TAPPING you can cut right-hand and left-hand threads.

No effect of the override controls during tapping
When Cycle 2.0 TAPPING is being run, the knobs for spindle speed override control and feed rate override control are disabled.

Required floating tap holder
A floating tap holder is required for executing Cycle 2.0 TAPPING. The floating tap holder compensates the tolerances for the programmed feed rate F and the programmed spindle speed S.

Tapping right-hand and left-hand threads
Right-hand thread: Spindle ON with miscellaneous function M 3
Left-hand thread: Spindle ON with miscellaneous function M 4

Process

The tapping cycle is illustrated in Fig. 7.4 and Fig. 7.5.

I:
The TNC pre-positions the tool at setup clearance Ⓐ above the workpiece surface.

II:
The tool drills to the end of thread Ⓑ at the feed rate F.

III:
When the tool reaches the end of thread, the direction of spindle rotation is reversed. After the programmed dwell time the tool is retracted to clearance height.

IV:
Above the workpiece, the direction of spindle rotation is reversed once again.

Calculating the feed rate F
Formula for calculation: **F = S • p** in [mm/min], where
S: Spindle speed in [rpm]
p: Pitch in [mm]

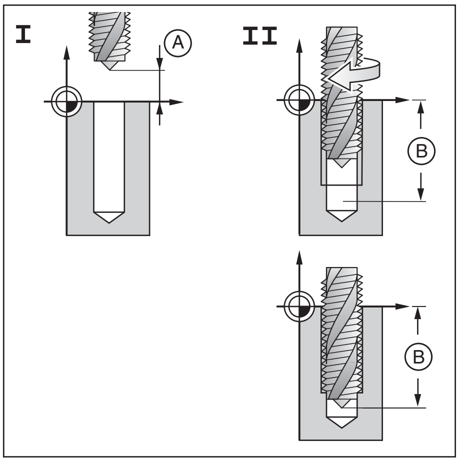


Fig. 7.4: Steps I and II in Cycle 2.0 TAPPING

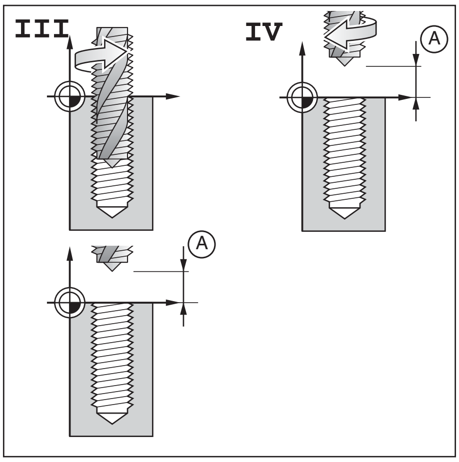
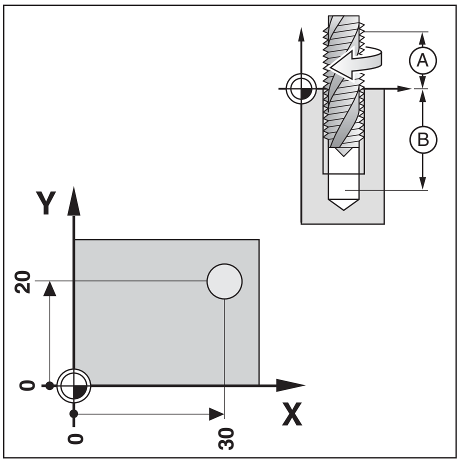


Fig. 7.5: Steps III and IV in Cycle 2.0 TAPPING

Input data for Cycle 2.0 TAPPING

- Clearance height - HEIGHT
  Position in the tool axis at which the TNC can move the tool in the working plane without damaging the workpiece.
- Setup clearance - DIST Ⓐ
  The TNC advances the tool from clearance height to setup clearance at rapid traverse.
  Standard value: DIST = 4 • thread pitch p
- Workpiece surface - SURF
  Absolute coordinate of the workpiece surface
- Thread length - DEPTH Ⓑ
  Distance between workpiece surface and end of thread.
- Dwell time - DWELL in [s]
  A dwell time prevents wedging of the tool when retracted.
  Further information is available from the machine manufacturer.
  Standard value: DWELL = 0 to 0.5 s
- Feed rate - F in [mm/min]
  Traversing speed of the tool during tapping

Programming example: Cycle `2.0 TAPPING`



Right-hand thread	
X coordinate of the hole:	30 mm
Y coordinate of the hole:	20 mm
Pitch p:	0.8 mm
Spindle speed `S`:	100 rpm
Clearance height `HEIGHT`:	+ 50 mm
Setup clearance `DIST` Ⓐ:	3 mm
Coordinate of the workpiece surface `SURF`:	0 mm
Thread depth `DEPTH` Ⓑ:	– 20 mm
Dwell time `DWELL`:	0.4 s
Feed rate `F = S` • p:	80 mm/min

Example: Entering Cycle `2.0 TAPPING` into a part program

Operating mode: `PROGRAMMING AND EDITING`

/	Page to the third soft-key row.
Cycle Def.	Select `Cycle Definition`.
Tapping	Enter Cycle `2.0 TAPPING` in a part program.
Clearance height ?	
5 0 ENT	Enter the clearance height (`HEIGHT = 50` mm). Confirm your entry.
Setup clearance ?	
3 ENT	Enter the setup clearance Ⓐ (`DIST = 3` mm). Confirm your entry.
Workpiece surface ?	
0 ENT	Enter the coordinate of the workpiece surface (`SURF = 0` mm). Confirm your entry.
Hole depth ?	
– 2 0 ENT	Enter the hole depth Ⓑ (`DEPTH = – 20` mm). Confirm your entry.

Dwell time ?	
0 • 4 ENT	Enter the dwell time (DWELL = 0.4 s). Confirm your entry.

Feed rate ?	
8 0 ENT	Enter the feed rate for tapping (F = 80 mm/min). Confirm your entry.

Program blocks

0	BEGIN PGM 30 MM	Start of program, program number, unit of measurement
1	F 9999	High feed rate for pre-positioning
2	Z+600	Tool-change position
3	X+30	Pre-positioning in the X axis
4	Y+20	Pre-positioning in the Y axis
5	TOOL CALL 4 Z	Call the tool for tapping, such as tool 4, tool axis Z
6	S 100	Spindle speed
7	M 3	Spindle ON, clockwise (**right-hand thread**)
8	**CYCL 2.0 TAPPING**	Cycle data for Cycle 2.0 TAPPING follow
9	**CYCL 2.1 HEIGHT +50**	Clearance height
10	**CYCL 2.2 DIST 3**	Setup clearance above the workpiece surface
11	**CYCL 2.3 SURF + 0**	Absolute coordinate of the workpiece surface
12	**CYCL 2.4 DEPTH -20**	Hole depth (thread length)
13	**CYCL 2.5 DWELL 0.4**	Dwell time at the end of thread
14	**CYCL 2.6 F 80**	Machining feed rate
15	CYCL CALL	Cycle call
16	M 2	Stop program run, spindle STOP, coolant OFF
17	END PGM 30 MM	End of program, program number, unit of measurement

Cycle 2.0 TAPPING is executed in the operating mode PROGRAM RUN (see Chapter 10).

Hole patterns in programs

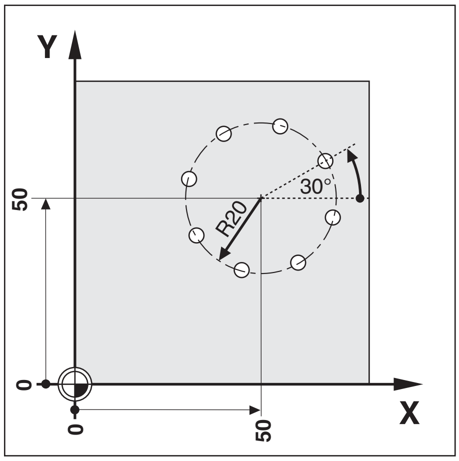
The information for the hole patterns `Circle Pattern` and `Linear Pattern` (see Chapter 4) can also be written to a program.

Executing holes in hole patterns
The TNC either drills bore holes or tap holes at the hole positions in the pattern. The bore hole or tap hole data, such as setup clearance and hole depth, must be programmed in a cycle.

The TNC then executes the holes according to the selected cycle that is programmed before the hole pattern cycle.

Hole pattern graphics
The hole patterns in a program can be displayed graphically.

Programming example: Cycle `5.0 Circle Pattern` (full circle)



Number of holes `NO.` :	8
Center point coordinates:	`CCX` = 50 mm
	`CCY` = 50 mm
Bolt circle radius `RAD`:	20 mm
Starting angle between X axis and first hole `START`:	30°

Hole data

A description of Cycle `1.0 Pecking` starts on page 79.

Clearance height `HEIGHT`:	+ 50 mm
Setup clearance `DIST`:	2 mm
Coordinate of the workpiece surface `SURF`:	0 mm
Hole depth `DEPTH`:	– 15 mm
Pecking depth `PECKG`:	5 mm
Dwell time `DWELL`:	0.5 s
Feed rate `F`:	80 mm/min

Example: Entering bolt hole circle data into a program

Operating mode: `PROGRAMMING AND EDITING`

/	Page to the third soft-key row.
Cycle Def.	Select `Cycle Definition`.
Circle Pattern	Select `Circle Pattern`. The soft-key row switches to a deeper level.

Type of bolt circle ?	
Full Circle	Select `Full Circle`. The TNC calculates the hole positions on a full circle.

Number of holes ?	
8 ENT	Enter the number of holes (`NO. = 8`). Confirm your entry.

Center point X ?	
5 0 ENT	Enter the X coordinate of the bolt circle center (`CCX = 50` mm). Confirm your entry.

Center point Y ?	
5 0 ENT	Enter the Y coordinate of the bolt circle center (`CCY = 50` mm). Confirm your entry.

Radius ?	
2 0 ENT	Enter the radius of the bolt circle (`RAD = 20` mm). Confirm your entry.

Starting angle ?	
3 0 ENT	Enter the starting angle from the X axis to the first hole (`START = 30°`). Confirm your entry.

Type of hole ?	
Pecking	Choose `Pecking` for drilling bore holes at the hole positions in the pattern.

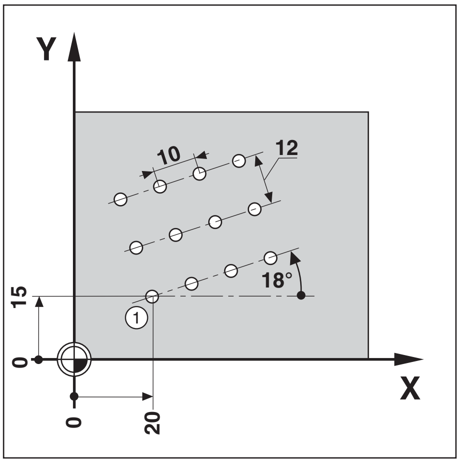
Program blocks

0	BEGIN PGM 40 MM	Start of program, program number, unit of measurement
1	F 9999	High feed rate for pre-positioning
2	Z+600	Tool-change position
3	TOOL CALL 3 Z	Call the tool for drilling, for example tool 3, tool axis Z
4	S 100	Spindle speed
5	M 3	Spindle ON, clockwise
6	**CYCL 1.0 PECKING**	Cycle data for Cycle 1.0 PECKING follow
7	**CYCL 1.1 HEIGHT +50**	Clearance height
8	**CYCL 1.2 DIST 2**	Setup clearance above the workpiece surface
9	**CYCL 1.3 SURF + 0**	Absolute coordinate of the workpiece surface
10	**CYCL 1.4 DEPTH -15**	Hole depth
11	**CYCL 1.5 PECKG 5**	Depth per infeed
12	**CYCL 1.6 DWELL 0.5**	Dwell time at bottom of hole
13	**CYCL 1.7 F 80**	Machining feed rate
14	**CYCL 5.0 FULL CIRCLE**	Cycle data for Cycle 5.0 FULL CIRCLE follow
15	**CYCL 5.1 NO. 8**	Number of holes
16	**CYCL 5.2 CCX +50**	X coordinate of the center of the bolt circle
17	**CYCL 5.3 CCY +50**	Y coordinate of the center of the bolt circle
18	**CYCL 5.4 RAD 20**	Radius
19	**CYCL 5.5 START +30**	Starting angle of first hole
20	**CYCL 5.6 TYPE 1:PECK**	Drill bore holes
21	M 2	Stop program run, spindle STOP, coolant OFF
22	END PGM 40 MM	End of program, program number, unit of measurement

For a **circle segment** (CYCL 6.0 CIRCLE SEGMENT) you also enter the angle step (STEP) between the holes (after the starting angle).

The bolt hole circle is then executed in the operating mode PROGRAM RUN (see Chapter 10).

Programming example: Cycle `7.0 Linear hole pattern`



X coordinate of the first hole ① :	`POSX` = 20 mm
Y coordinate of the first hole ① :	`POSY` = 15 mm
Number of holes per row `NO.HL`:	4
Hole spacing `HLSPC`:	10 mm
Angle between hole row and X axis `ANGLE`:	18°
Number of rows `NO.RW`:	3
Row spacing `RWSPC`:	12 mm

Hole data

A description of Cycle `1.0 Pecking` starts on page 79.

Clearance height `HEIGHT`:	+ 50 mm
Setup clearance `DIST`:	2 mm
Coordinate of the workpiece surface `SURF`:	0 mm
Hole depth `DEPTH`:	– 15 mm
Infeed depth `PECKG`:	5 mm
Dwell time `DWELL`:	0.5 s
Feed rate `F`:	80 mm/min

Example: Entering data for linear hole pattern into a program

Operating mode: `PROGRAMMING AND EDITING`

/	Page to the third soft-key row.
Cycle Def.	Select `Cycle Definition`.
Linear Pattern	Select `Linear Pattern`.

Dialog	Input	Description
`1st hole X ?`	2 0 ENT	Enter the X coordinate of hole ① (`POSX = 20` mm). Confirm your entry.
`1st hole Y ?`	1 5 ENT	Enter the Y coordinate of hole ① (`POSY = 15` mm). Confirm your entry.
`Holes per row ?`	4 ENT	Enter the number of holes per row (`NO.HL = 4`). Confirm your entry.
`Hole spacing ?`	1 0 ENT	Enter the hole spacing (`HLSPC = 10` mm). Confirm your entry.
`Angle ?`	1 8 ENT	Enter the angle between the X axis and the rows of holes (`ANGLE = 18°`). Confirm your entry.
`Number of rows ?`	3 ENT	Enter the number of rows (`NO.RW = 3`). Confirm your entry.
`Row spacing ?`	1 2 ENT	Enter the row spacing (`RWSPC = 12` mm). Confirm your entry.
`Type of hole ?`	Pecking	Choose `Pecking` for drilling bore holes at the hole positions in the pattern.

Program blocks

0	BEGIN PGM 50 MM	Start of program, program number, unit of measurement
1	F 9999	High feed rate for pre-positioning
2	Z+600	Tool-change position
3	TOOL CALL 5 Z	Call the tool for pecking, such as tool 5, tool axis Z
4	S 1000	Spindle speed
5	M 3	Spindle ON, clockwise
6	**CYCL 1.0 PECKING**	Cycle data for Cycle `1.0 PECKING` follow
7	**CYCL 1.1 HEIGHT +50**	Clearance height
8	**CYCL 1.2 DIST 2**	Setup clearance above the workpiece surface
9	**CYCL 1.3 SURF + 0**	Absolute coordinate of the workpiece surface
10	**CYCL 1.4 DEPTH -15**	Hole depth
11	**CYCL 1.5 PECKG 5**	Depth per infeed
12	**CYCL 1.6 DWELL 0.5**	Dwell time at bottom of hole
13	**CYCL 1.7 F 80**	Machining feed rate
14	**CYCL 7.0 LINEAR HOLE PATTN**	Cycle data for Cycle `7.0 LINEAR HOLE PATTN` follow
15	**CYCL 7.1 POSX +20**	X coordinate of first hole 1
16	**CYCL 7.2 POSY +15**	Y coordinate of first hole 1
17	**CYCL 7.3 NO.HL 4**	Number of holes per row
18	**CYCL 7.4 HLSPC +10**	Distance between holes on the row
19	**CYCL 7.5 ANGLE +18**	Angle between the rows and the X axis
20	**CYCL 7.6 NO.RW 3**	Number of rows
21	**CYCL 7.7 RWSPC +12**	Spacing between rows
22	**CYCL 7.8 TYPE 1:PECK**	Pecking
23	M 2	Stop program run, spindle STOP, coolant OFF
24	END PGM 50 MM	End of program, program number, unit of measurement

The hole pattern is then executed in the operating mode `PROGRAM RUN` (see Chapter 10).

Rectangular pockets in programs

The TNC makes it easier to clear out rectangular pockets. You need only enter the dimensions of the pocket; the TNC calculates the tool path for you.

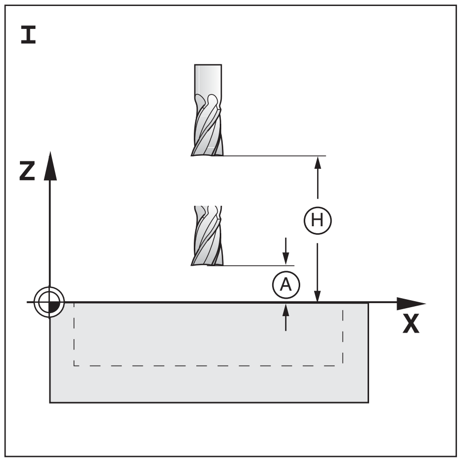


Fig. 7.6: Step `I` in Cycle `4.0 RECTANGULAR POCKET`

Process

The cycle process is illustrated in Figures 7.6, 7.7 and 7.8.

I:
The TNC pre-positions the tool in the tool axis at the clearance height H , moves it in the working plane to the pocket center, then in the tool axis to the setup clearance A.

II:
The TNC drills at the pecking feed rate to the first pecking depth C.

III:
The TNC clears out the pocket at the milling feed rate along the path illustrated in Fig. 7.8 below (in this case with climb milling).

IV:
The pecking and the roughing process are repeated down to the programmed depth B . Then the TNC ends the cycle by moving the tool in the pocket center back to the clearance height H .

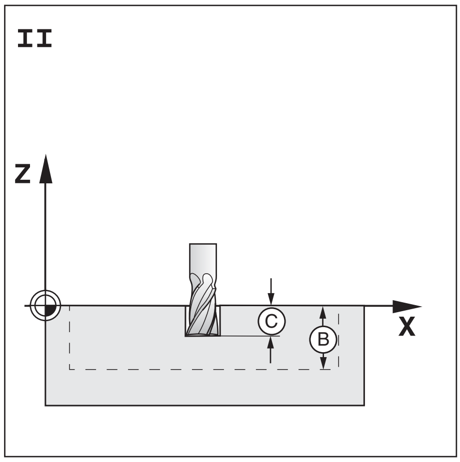


Fig. 7.7: Step `II` in Cycle `4.0 RECTANGULAR POCKET`

Input data for Cycle `4.0 RECTANGULAR POCKET`

- Clearance height — `HEIGHT` H
  The absolute position in the tool axis at with the tool can move in the working plane without danger of collision.
- Setup clearance — `DIST` A
  The tool moves at rapid traverse from the clearance height to the setup clearance.
- Workpiece surface — `SURF`
  Absolute coordinate of the workpiece surface.
- Milling depth — `DEPTH` B
  Distance between workpiece surface and bottom of pocket.
- Pecking depth — `PECKG` C
  Infeed per drilling cut.
- Pecking feed rate — `F`
  Tool traversing speed during pecking.
- Pocket center in X — `POSX` MX
  Point in the longitudinal axis at which the pocket center is located.
- Pocket center in Y — `POSY` MY
  Point in the transverse axis at which the pocket center is located.
- Side length in X — `LNGTH X` X
  Length of the pocket in the longitudinal axis.
- Side length in Y — `LNGTH Y` Y
  Length of the pocket in the transverse axis.
- Milling feed rate — `F`
  Traversing speed of the tool in the working plane.
- Direction `DIRCTN`
  Input value 0: climb milling (Fig. 7.8: clockwise)
  Input value 1: upcut milling (counterclockwise)
- Finishing allowance - `ALLOW`
  Finishing allowance in the working plane.

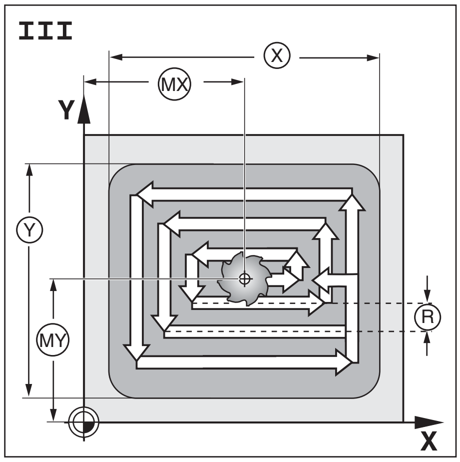
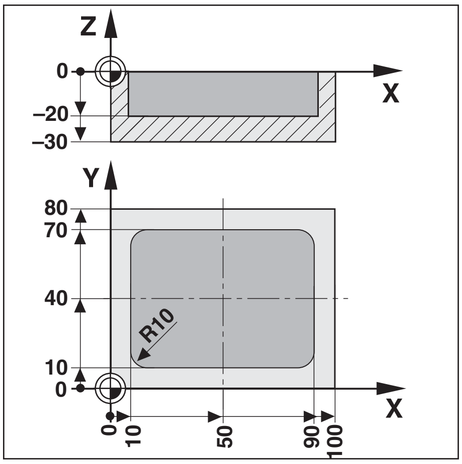


Fig. 7.8: Step `III` in Cycle `4.0 RECTANGULAR POCKET`

Example: Cycle `4.0 RECTANGULAR POCKET`

Clearance height:	+ 80	mm
Setup clearance:	2	mm
Workpiece surface:	+ 0	mm
Milling depth:	– 20	mm
Pecking depth:	7	mm
Pecking feed rate:	80	mm/min
Pocket center in X:	50	mm
Pocket center in Y:	40	mm
Side length in X:	80	mm
Side length in Y:	60	mm
Milling feed rate:	100	mm/min
Direction:	0: CLIMB	
Finishing allowance:	0.5	mm



Example: Entering Cycle `4.0 RECTANGULAR POCKET` into a part program

Operating mode: `PROGRAMMING AND EDITING`

/	Page to the third soft-key row.
Cycle Def.	Select `Cycle Definition`.
Pocket Milling	Enter Cycle `4.0 RECTANGULAR POCKET` in a part program.
Clearance height ?	
8 0 ENT	Enter the clearance height (`HEIGHT = 80` mm). Confirm your entry.
Setup clearance ?	
2 ENT	Enter the setup clearance (`DIST = 2` mm). Confirm your entry.
Workpiece surface ?	
0 ENT	Enter the coordinate of the workpiece surface (`SURF = 0` mm). Confirm your entry.

.
.
.

Program blocks

0	BEGIN PGM 55 MM	Start of program, program number, unit of measurement
1	F 9999	High feed rate for pre-positioning
2	Z+600	Tool-change position
3	X-100	Pre-positioning in the X axis
4	Y-100	Pre-positioning in the Y axis
5	TOOL CALL 7 Z	Call the tool for pocket milling, such as tool 7, tool axis Z
6	S 800	Spindle speed
7	M 3	Spindle ON, clockwise
8	**CYCL 4.0 RECTANGULAR POCKET**	Cycle data for Cycle `4.0 RECTANGULAR POCKET` follow
9	**CYCL 4.1 HEIGHT + 80**	Clearance height
10	**CYCL 4.2 DIST 2**	Setup clearance above the workpiece surface
11	**CYCL 4.3 SURF + 0**	Absolute coordinate of the workpiece surface
12	**CYCL 4.4 DEPTH – 20**	Milling depth
13	**CYCL 4.5 PECKG 7**	Depth per infeed
14	**CYCL 4.6 F 80**	Pecking feed rate
15	**CYCL 4.7 POSX + 50**	Pocket center in X
16	**CYCL 4.8 POSY + 40**	Pocket center in Y
17	**CYCL 4.9 LNGTHX 80**	Side length X
18	**CYCL 4.10 LNGTHY 60**	Side length Y
19	**CYCL 4.11 F 100**	Milling feed rate
20	**CYCL 4.12 DIRCTN 0: CLIMB**	Climb milling
21	**CYCL 4.13 ALLOW 0.5**	Finishing allowance
22	M 2	Stop program run, spindle STOP, coolant OFF
23	END PGM 55 MM	End of program, program number, unit of measurement

Cycle `4.0 RECTANGULAR POCKET` is executed in the operating mode `PROGRAM RUN` (see Chapter 10).

8
Subprograms and Program Section Repeats

Subprograms and program section repeats only need to be entered once in the program. You can then run them up to 999 times.

Subprograms can be run at any point in the program, while program section repeats are run several times in succession.

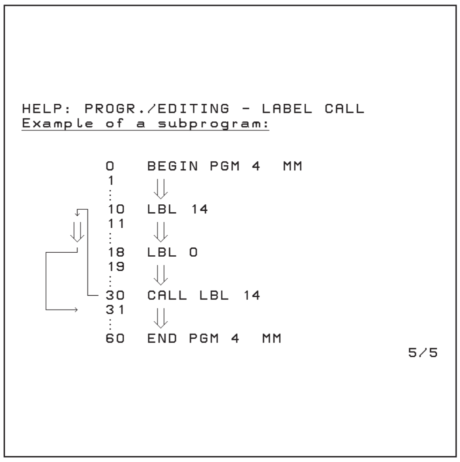


Fig. 8.1: On-screen operating instructions for subprogram *(page 5 shown)*

Inserting program marks (labels)

You identify subprograms and program section repeats with labels (abbreviated in the program to `LBL`).

Labels 1 to 99
Labels 1 to 99 identify the beginning of a subprogram or a program section which is to be repeated.

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

Label call
In the program, subprograms and program sections are called with the command `CALL LBL`.
The command `CALL LBL 0` is not allowed.
Subprograms:
After a `CALL LBL` block in the program, the TNC executes the called subprogram.
Program section repeats:
The TNC repeats the program section located before the `CALL LBL` block. You enter the number of repeats with the `CALL LBL` command.

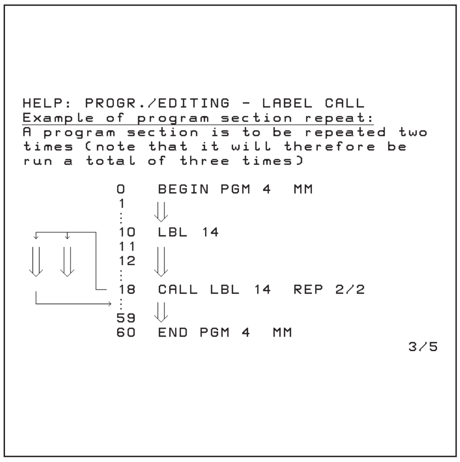


Fig. 8.2: On-screen operating instructions for program section repeats *(page 3 shown)*

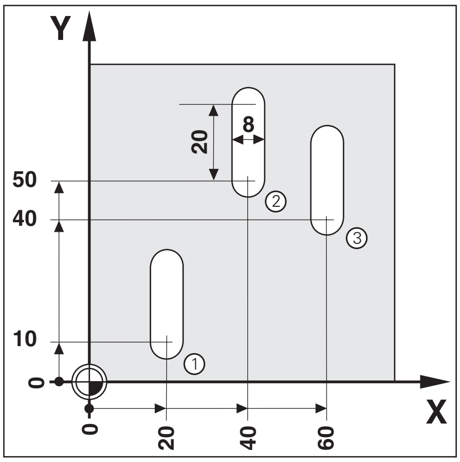
Nesting program sections

Subprograms and program section repeats can also be “nested.” For example, a subprogram can in turn call another subprogram.

Maximum nesting depth: 8 levels

Subprograms

Programming example: Subprogram for slots



Slot lengths: 20 mm + tool diameter
Slot depths: – 10 mm
Slot diameters: 8 mm (= tool diameter)
Infeed point coordinates
Slot ① : X = 20 mm Y = 10 mm
Slot ② : X = 40 mm Y = 50 mm
Slot ③ : X = 60 mm Y = 40 mm

> ⚠ This example requires a center-cut end mill (ISO 1641)!

Example: Inserting label for subprogram

Operating mode: `PROGRAMMING AND EDITING`

/	Go to the second soft-key row.
Label Number	Insert a label (`LBL`) for a subprogram. The TNC offers the lowest available number.

Label number ?

ENT	Accept the default label number.
or	or
1 ENT	Enter a label number (here, 1). Confirm your entry. The current block now contains the label `LBL 1`.

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 (`LBL 0`) is used **only** to identify the **end** of a subprogram.

Example: Entering a subprogram call: `CALL LBL`

/	Go to the second soft-key row.
Label Call	Call label. The TNC offers the label number which was last set.

Label number ?	
ENT	Accept the default label number.
or	or
1 ENT	Enter a label number (here, 1). Confirm your entry. The current block now contains the called label: `CALL LBL 1`.
Sub-program	For subprograms you can ignore the question "`Repeat REP ?`". Press the soft key to confirm that a subprogram is being called.

After the `CALL LBL` block in the operating mode `PROGRAM RUN`, the TNC executes those blocks in the subprogram that are located between the `LBL` block with the called number and the next block containing `LBL 0`.

Note that the subprogram will be executed at least once even without a `CALL LBL` block.

Program blocks

0	BEGIN PGM 60	MM	Start of program, program number, unit of measurement
1	F 9999		High feed rate for pre-positioning
2	Z+20		Clearance height
3	X+20	R0	X coordinate infeed point slot 1
4	Y+10	R0	Y coordinate infeed point slot 1
5	TOOL CALL 7 Z		Call tool data, here tool 7, tool axis Z
6	S 1000		Spindle speed
7	M 3		Spindle ON, clockwise
8	CALL LBL 1		Call subprogram 1: execute blocks 17 to 23
9	X+40	R0	X coordinate infeed point slot 2
10	Y+50	R0	Y coordinate infeed point slot 2
11	CALL LBL 1		Call subprogram 1: execute blocks 17 to 23
12	X+60	R0	X coordinate infeed point slot 3
13	Y+40	R0	Y coordinate infeed point slot 3
14	CALL LBL 1		Call subprogram 1: execute blocks 17 to 23
15	Z+20		Clearance height
16	M 2		Stop program run, spindle STOP, coolant OFF
17	**LBL 1**		Start of subprogram 1
18	**F 200**		Machining feed rate during subprogram
19	**Z-10**		Infeed to slot depth
20	**IY+20**	**R0**	Mill slot
21	**F 9999**		High feed rate for retracting and pre-positioning
22	**Z+2**		Retract
23	**LBL 0**		End of subprogram 1
24	END PGM 60	MM	End of program, program number, unit of measurement

Program section repeats

A program section repeat is entered like a subprogram. The end of the program section is identified simply by the command to repeat the section.

Label 0 is therefore not set.

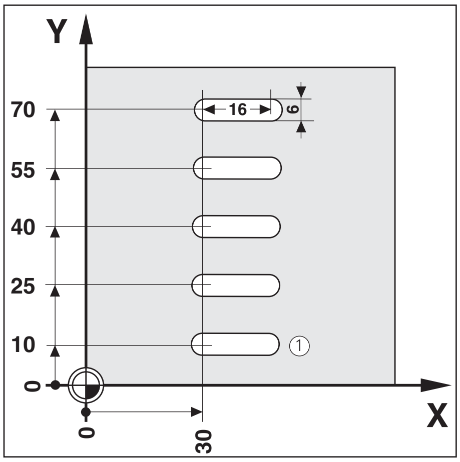
Display of the `CALL LBL` block with a program section repeat

The screen displays (for example): `CALL LBL 1 REP 10 / 10 .`

The two numbers with the slash between them indicate that this is a program section repeat. The number **in front of** the slash is the number of repeats you entered. The number **behind** the slash is the number of repeats remaining to be performed.

Programming example: Program section repeat for slots

Slot lengths: 16 mm + tool diameter
Slot depths: – 12 mm
Incremental offset
of the infeed point : 15 mm
Slot diameter: 6 mm (= tool diameter)
Infeed point coordinates
Slot ① : X = 30 mm Y = 10 mm



> ⚠ This example requires a center-cut end mill (ISO 1641)!

Example: Label for a program section repeat

Operating mode: `PROGRAMMING AND EDITING`

/	Go to the second soft-key row.
Label Number	Insert a label for a program section repeat (`LBL`). The TNC offers the lowest available label number as a default.
Label number ?	
ENT	Accept the default label number.
or	or
1 ENT	Enter a label number (here, 1). Confirm entry. The current block now contains the set label: `LBL 1`.

Enter the blocks for the program section repeat after the `LBL` block.

Example: Entering a program section repeat: `CALL LBL`

/	Go to the second soft-key row.
Label Call	Call label. The TNC offers the label number that was last set.
Label number ?	
ENT	Accept the default label number.
or	or
1 ENT	Enter a label number (here, 1). Confirm your entry. The current block now contains the called label: `CALL LBL 1`.
Repeat REP ?	
4 ENT	Enter the number of repeats (here, 4). Confirm your entry.

After a `CALL LBL` block in the operating mode `PROGRAM RUN`, the TNC repeats those program blocks that are located **behind** the LBL block with the called number and **before** the `CALL LBL` block.

Note that the program section will always be executed one more time than the programmed number of repeats.

Program blocks

0	BEGIN PGM 70	MM	Start of program, program number, unit of measurement
1	F 9999		High feed rate for pre-positioning
2	Z+20		Clearance height
3	TOOL CALL 9 Z		Call tool data, here tool 9, tool axis Z
4	S 1800		Spindle speed
5	M 3		Spindle ON, clockwise
6	X+30	R0	X coordinate infeed point slot 1
7	Y+10	R0	Y coordinate infeed point slot 1
8	**LBL 1**		Start of program section 1
9	**F 150**		Machining feed rate during program section repeat
10	**Z-12**		Infeed
11	**IX+16**	**R0**	Mill slot
12	**F 9999**		High feed rate for retracting and pre-positioning
13	**Z+2**		Retract
14	**IX-16**	**R0**	Positioning in X
15	**IY+15**	**R0**	Positioning in Y
16	**CALL LBL 1 REP 4 / 4**		Repeat program section 1 four times
17	Z+20		Clearance height
18	M 2		Stop program run, spindle STOP, coolant OFF
19	END PGM 70	MM	End of program, program number, unit of measurement

NOTES

9
Transferring Files Over the Data Interface

The TNC 124 features an RS-232-C interface for external data storage on a device such as the HEIDENHAIN FE 401 floppy disk unit or a PC.
Programs, tool tables and datum tables can also be archived on diskette and loaded back into the TNC again as required.

> Pin layout, wiring and connections for the data interface are described on page 115 and in the Technical Manual for the TNC 124.

Functions for data transfer

Function	Soft key/Key
Directory of programs stored in the TNC	TNC 124 PGM Dir
Directory of programs stored on the FE	FE 401 PGM Dir
Abort data transfer	Escape
• Toggle between FE and EXT • Show further programs	→

Transferring a program into the TNC

Operating mode: PROGRAMMING AND EDITING

Program Manage.	Select Program Manage.
Extern.	Select Extern.
File number ?	
5	Enter the program number (here, 5).
→	Select external device (for diskette unit or PC with HEIDENHAIN data transfer software TNC.EXE use FE setting; for PC without TNC.EXE use EXT setting).
Start Input	Press Start Input to transfer the program to the TNC. The message Loading file: appears on the TNC screen.

If you are **transferring programs from a PC** into the TNC (EXT setting), the PC must **send** the programs.

Reading a program out of the TNC

Example: Reading a program out of the TNC

Operating mode: PROGRAMMING AND EDITING

Program Manage.	Select Program Manage.
Extern.	Select Extern.
⬚ / ⬚	Go to EXTERNAL OUTPUT.
File number ?	
1 0	Enter the program number (here, 10).
→	Select the external device. For diskette unit or PC with HEIDENHAIN data transfer software TNC.EXE use FE setting; for PC without TNC.EXE or printer use EXT setting.
Start Output	Press Start Output to transfer the program to the external device. The message Reading out file: appears on the TNC screen.

CAUTION

A program on the external device with the same number as that being read out will be overwritten. No confirmation to overwrite will be requested!

To read all programs out of the TNC:

If you wish to read all programs out of the TNC:

➤ Press the soft key Output All PGM.

Transferring tool tables and datum tables

Operating mode: `any`

MOD	Select the user parameters.
Tool Table **or** Datum Table	Choose tool table or datum table.
/	Go to the second soft-key row.
File number?	
2	Enter the number of the tool table or datum table you wish to transfer.
Start Input **or** Start Output	Start data input or start data output.

10
Executing programs

Programs are run in the operating mode `PROGRAM RUN`.

There are two ways to run programs:

`Single block`
Use the NC– `I` key to start the current program block (displayed between the dashed lines on the TNC screen). It is recommended that you use `Single block` when running a program for the first time.

`Automatic`
The TNC automatically executes the program block by block until program run is interrupted or execution of the program has been completed. Use `Automatic` when you are sure the program contains no errors and you want to run it quickly.

Pre-positioning the tool

Before running a part program, always pre-position the tool to prevent the possibility of damaging the tool or workpiece. The best pre-position lies outside the programmed contour on the extension of the tool path for machining the first contour point.

Sequence in which the tool approaches the pre-position for milling

➤ Insert the tool at clearance height.
➤ Move the tool in X and Y (tool axis Z) to the pre-position coordinates.
➤ Move the tool to the working depth.

Preparation

➤ Clamp the workpiece to the machine table.
➤ Select the desired datum point (see "Selecting datum points").
➤ Set the workpiece datum.
➤ Press `Program Number` to select the program you want to execute.

Changing the feed rate F and spindle speed S during program run

During program run, you can vary the feed rate F and the spindle speed S infinitely from 0% to 150% of the set values by turning the override knobs on the TNC control panel.

> Some TNCs **do not** have a knob for spindle speed override.

Overview of functions

Function	Soft key/Key
Start with the block before the current block	↑
Start with the block after the current block	↓
Select the starting block directly	GOTO □
Stop machine axis movements; Interrupt program run	NC 0
Abort program run	INTERN-STOP
Enter the tool data	Tool Table
Single Block: Skip program blocks	Next Block

Single block

Operating mode: PROGRAM RUN

If required: Single Block	If PROGRAM RUN FULL SEQUENCE is displayed at the top of the screen, select Single Block.
For each block: NC I	Position for each individual program block.

Continue positioning and calling blocks with the NC–I key until machining is complete.

Skipping program blocks

The TNC can skip blocks in the operating mode PROGRAM RUN SINGLE BLOCK.

To skip a program block:

➤ Press the soft key Next Block.

Move the machine axes **directly** to the position that is displayed as the current block (the TNC accounts for incremental positions from skipped blocks):

➤ Press the NC–I key.

Full sequence

> The machine manufacturer determines whether or not the `Full Sequence` function is enabled on your TNC.

Operating mode: `PROGRAM RUN`

If required: `Full Sequenc`	If `PROGRAM RUN SINGLE BLOCK` is displayed at the top of the screen, go to `Automatic`.
NC I	Position.

The TNC automatically executes the next positioning block as soon as it has reached the programmed position.

Interrupting program run

Interrupt program run, but **do not abort**:

➤ Press NC–0.

To resume program run after an interruption:

➤ Press NC–I.

Stop program run and **abort**:

➤ Press NC–0.
The soft-key row contains the soft key `INTERN. STOP`.

➤ Press `INTERN. STOP`.

To restart program run after STOP

The TNC interrupts program run as soon as it reaches a `STOP` block.

To restart program run:

➤ Press NC–I.

11

Positioning Non-Controlled Axes

The machine manufacturer determines which axes will be controlled automatically by the TNC and which will be controlled via mechanical handwheels.

The machine manufacturer sets two operating modes for the display of non-controlled axes:

- The position display shows the actual position of the machine slide
- The position display shows the distance-to-go to the programmed nominal position

You will recognize an axis in distance-to-go mode by the Δ symbol to the upper right of the axis designation.

If your TNC displays distance-to-go to nominal position, you can program and execute a manually traversed axis simply by moving the machine slide to display value zero.

The distance-to-go mode functions as follows during Program Run:

➤ Enter the program including the manual positionings.
➤ Start program run.
➤ The TNC will stop program run at manual positioning blocks.
➤ Position the machine slide manually by traversing to zero.
➤ Start program run once again.

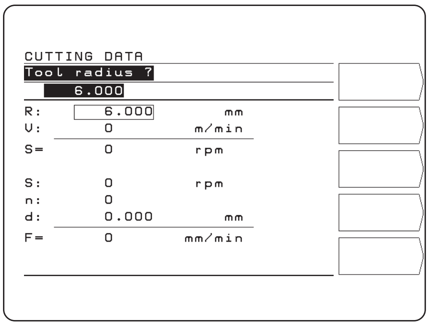
12
Cutting Data Calculator, Stopwatch and Pocket Calculator: The INFO Functions

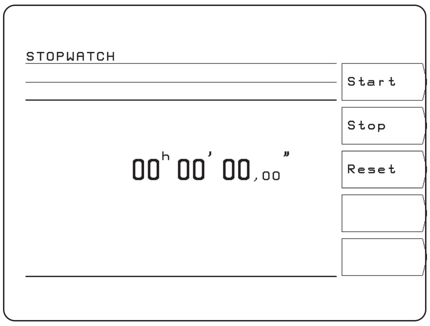
Press the INFO key to access the following functions:

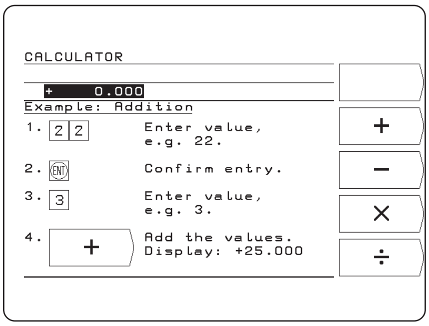
- **Cutting data calculator**
  Calculates the spindle speed from the tool radius and the cutting speed;
  Calculates the feed rate from the spindle speed, the number of teeth and the depth of cut per tooth.
- **Stopwatch**
- **Pocket calculator**
  Basic arithmetic + , − , x , ÷
  Trigonometric functions (sin, cos, tan, arc sin, arc cos, arc tan)
  Square roots
  x^2
  Reciprocals (1/x)
  π (3.14159...)

To access the INFO functions

INFO	Press the INFO key.
`Cutting Data`	**`Cutting Data`** for milling.
or	or
`Stop-watch`	Select **`Stopwatch`**.
or	or
`Calc.`	Select **`Calculator`** functions.







Cutting data: Calculate spindle speed S and feed rate F

The TNC can calculate the spindle speed S and the feed rate F for you. As soon as you conclude an entry with ENT, the TNC prompts you for the next entry.

Entry values

- For the spindle speed S in rpm:
  Enter the tool radius R in mm and the cutting speed V in m/min.
- For the feed rate F in mm/min:
  Enter the spindle speed S in rpm,
  the number of teeth *n* of the tool and
  the permissible depth of cut per tooth *d* in mm.

For calculation of the feed rate, the TNC automatically offers the spindle speed it just calculated. You can enter a different value, however.

Overview of functions

Function	Key
Confirm entry and continue dialog	ENT
Go to the next-higher input line	↑
Go to the next-lower input line	↓

Example: Entering the tool radius

You can be in any operating mode. Select `Cutting Data`.

Tool radius ?	
8 ENT	Enter the tool radius (8 mm) and transfer it into the box behind the letter R.

Stopwatch

The stopwatch shows the hours (h), minutes (‘), seconds (‘‘) and hundredths of a second.
The stopwatch function continues to run even when you leave the INFO function. When the power is interrupted (switch-off), the TNC resets the stopwatch to zero.

Function	Soft key
Start timing	Start
Stop timing	Stop
Reset the stopwatch	Reset

Pocket calculator functions

The pocket calculator functions are spread over three soft-key rows:

- Basic arithmetic (first soft-key row)
- Trigonometry (second row)
- Square root, x^2, 1/x, π (third row)

Use the paging keys to go from one soft-key row to the next.
The TNC always shows an example entry.

Transferring the calculated value

The calculated value remains in the input line even after you leave the pocket calculator function.
This allows you to transfer the calculated value directly into a program as a nominal position — without having to reenter it.

Entry logic

For calculations with **two** operands (addition, subtraction, etc.):

➤ Key in the first value.
➤ Confirm the value by pressing ENT.
➤ Key in the second value.
➤ Press the soft key for the desired operation.
The TNC displays the result of the operation in the input line.

For calculations with **one** operand (sine, reciprocal, etc.):

➤ Key in the value.
➤ Press the soft key for the desired operation.
The TNC displays the result of the operation in the input line.

Example: See the next page.

Example: (3x4+14) ÷(2x6+1) = 2

3 ENT	Key in the first value in the first parenthesis: 3; confirm entry. The display shows `+3.000`.
4 ×	Key in the second value in the first parenthesis: 4 and combine the second value with the first value: x. The display now shows `+12.000`.
1 4 +	Key in the third value in the first parenthesis: 14 and combine the third value with the displayed value 12.000: +. The display now shows `+26.000`.
2 ENT	Key in the first value in the second parenthesis: 2; confirm entry. This automatically closes the first parenthesis. The display shows `+2.000`.
6 ×	Key in the second value in the second parenthesis: 6 and combine the second value with the first value: x. The display now shows `+12.000`.
1 +	Key in the third value in the second parenthesis: 1 and combine the third value with the displayed value 12.000: +. The display now shows `+13.000`.
÷	Close the second parenthesis and simultaneously combine with the first parenthesis: ÷. The display now shows the result: `+2.000`.

13
User Parameters: The MOD Function

User parameters are operating parameters which you can change without having to enter a code number. The machine manufacturer determines which operating parameters are available to you as user parameters as well as how the user parameters are arranged in the soft keys.

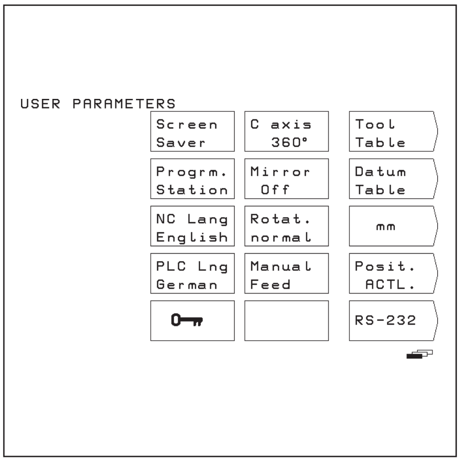


Fig. 12.1: The user parameters on the TNC screen

To access the user parameter menu

➤ Press MOD.
The user parameters appear on the screen.
➤ Page to the soft-key row containing the desired user parameter.
➤ Press the soft key for the desired user parameter.

To leave the user parameter menu

➤ Press MOD.

Entering user parameters

Choosing settings

Some user parameter settings are chosen directly with the soft keys. You simply switch from one setting to another.

Example: Scaling factor

➤ Press MOD.

➤ Go to the soft-key row containing `mm` or `inch`

➤ Press the displayed soft key.
The soft key changes to the other setting, for example from `mm` to `inch`.
The displayed setting is active.

➤ Press MOD again.
This ends the MOD function.
The new setting for the angle format is now in effect.

Changing settings

Some user parameters require that you enter a value and confirm your entry with ENT.

Example: User parameter for screen saver.

MOD

TNC 124 user parameters

Parameter	Soft key	Settings/Comments
Position display	`Posit.`	ACTL., NOML., REF, LAG
Unit of measurement	`mm` `inch`	Dimensions in mm Dimensions in inches
Display mode Rotary axis	`.. axis`	0 to 360° –180° to 180° ∞
Tool table	`Tool Table`	Edit tool table and select tools
Datum table	`Datum Table`	Select and edit datum points
Data transfer rate (baud rate)	`RS-232`	300, 600, 1 200, 2 400 baud 4 800, 9 600, 38 400 baud
Bolt hole circle graphic	`Rotat.`	Normal (positive counterclockwise) Inverse
Linear hole pattern graphic	`Mirror`	Off Vert.: Mirror vertically Horiz.: Mirror horizontally Ve+Hor.: Mirror vertically and horizontally
Feed rate manual operation	`F`	Feed rate during traverse with the direction keys
Dialog language	`NC`	German English
PLC dialog language	`PLC`	German, English, French, Italian, Spanish, ...
Screen saver	`Sleep`	5 to 98 [min] Off = 99
Programming station	`Progrm. Station`	TNC with machine Programming station with PLC Programming station without PLC
Code number	(Code)	Change operating parameters that are not user parameters
Marker	`Marker ...`	Machine-dependent function

14
Tables, Overviews and Diagrams

This chapter contains information which you will frequently need when working with the TNC:

- Overview of miscellaneous functions (M functions) with predetermined effect
- Overview of vacant miscellaneous functions
- Diagram for determining the feed rate for tapping
- Technical information
- Overview of accessories

Miscellaneous functions (M functions)

Miscellaneous functions with predetermined effect

With the miscellaneous functions the TNC particularly controls:

- Coolant (ON/OFF)
- Spindle rotation (ON/OFF/direction of rotation)
- Program run
- Tool change

The machine manufacturer determines which miscellaneous functions are available on your TNC and which functions they have.

M no.	Standard miscellaneous function
M00	Stop program run, spindle STOP, coolant OFF
M02	Stop program run, spindle STOP, coolant OFF, go to block 1
M03	Spindle ON, clockwise
M04	Spindle ON, counterclockwise
M05	Spindle STOP
M06	Tool change, stop program run, spindle STOP
M08	Coolant ON
M09	Coolant OFF
M13	Spindle ON, clockwise, coolant ON
M14	Spindle ON, counterclockwise, coolant ON
M30	Stop program run, spindle STOP, coolant OFF, go to block 1

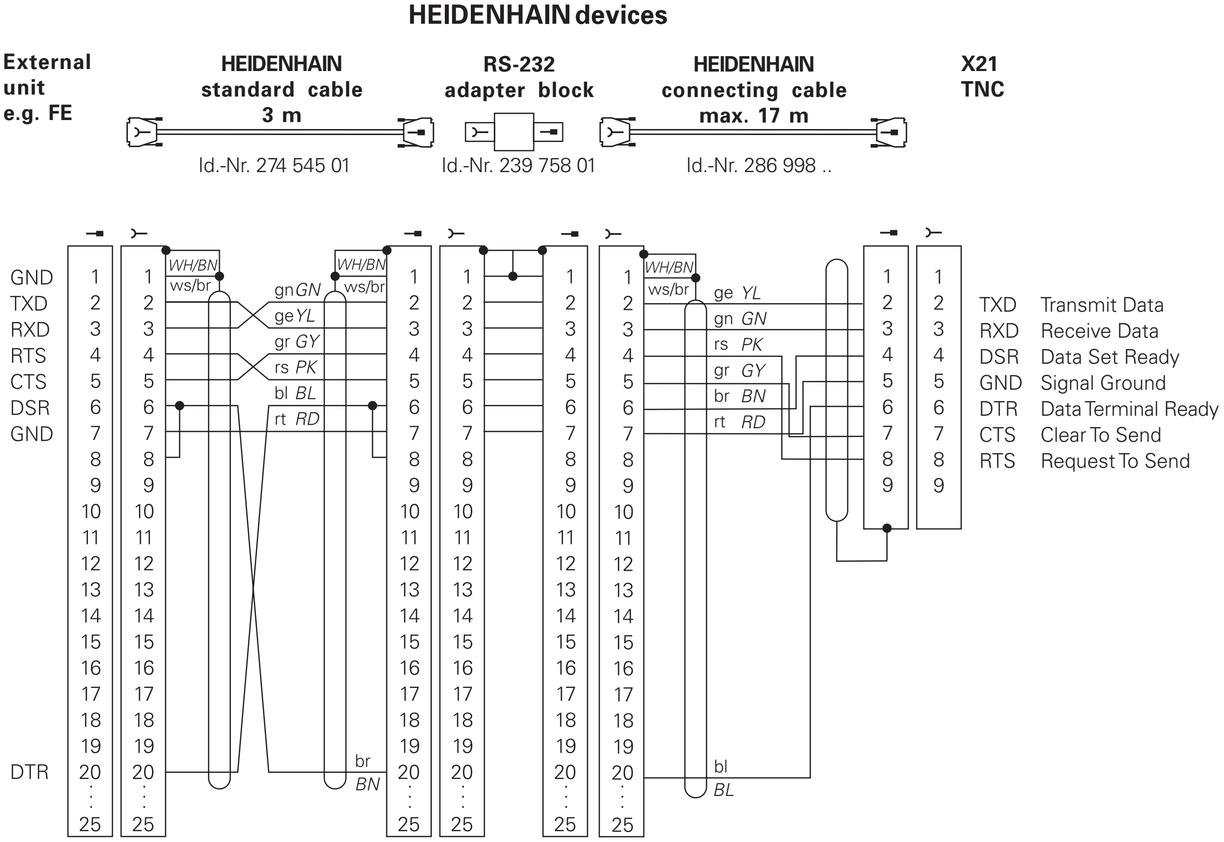
Vacant miscellaneous functions

The machine manufacturer can provide you with information on the machine-specific functions he has assigned to the vacant miscellaneous functions listed on this page.

M number	Vacant M function
M01	
M07	
M10	
M11	
M12	
M15	
M16	
M17	
M18	
M19	
M20	
M21	
M22	
M23	
M24	
M25	
M26	
M27	
M28	
M29	
M31	
M32	
M33	
M34	
M35	
M36	
M37	
M38	
M39	
M40	
M41	
M42	
M43	
M44	
M45	
M46	
M47	
M48	
M49	

M number	Vacant M function
M50	
M51	
M52	
M53	
M54	
M55	
M56	
M57	
M58	
M59	
M60	
M61	
M62	
M63	
M64	
M65	
M66	
M67	
M68	
M69	
M70	
M71	
M72	
M73	
M74	
M75	
M76	
M77	
M78	
M79	
M80	
M81	
M82	
M83	
M84	
M85	
M86	
M87	
M88	
M89	

Pin layout and connecting cable for the data interface



The connector pin layout on the adapter block differs from that on the TNC logic unit (X 21).

The X21 interface complies with "safe separation from line power" as required by EN 50 178.

Connecting non-HEIDENHAIN devices

The connector pin layout on a non-HEIDENHAIN device may be quite different from that on a HEIDENHAIN device. This depends on the unit and the type of data transfer.

Diagram for machining

> The TNC calculates the spindle speed S and feed rate F with the INFO function `Cutting Data` (see Chapter 12).

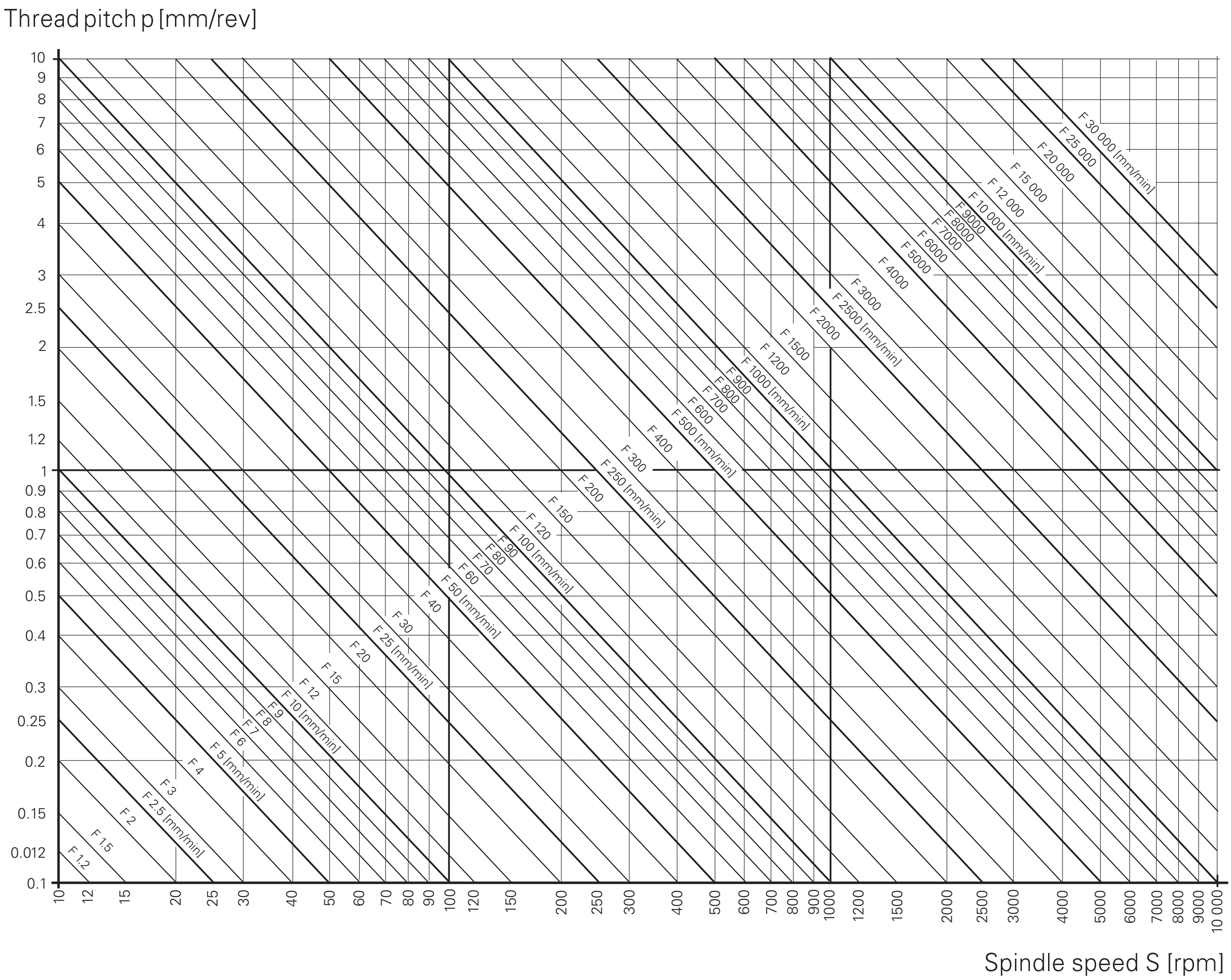
Feed rate F for tapping

F = p • S [mm/min]

F: Feed rate in [mm/min]
p: Thread pitch [mm]
S: Spindle speed in [rpm]

Example: Calculating the feed rate F for tapping

p = 1 mm/rev
S = 500 rpm
F = 100 mm/min (F = 100 mm/min as read from the diagram)



Technical information

TNC data	
Brief description	Contouring control with analog speed control for machines with up to 4 axes (axis control for up to 3 axes, position display for a fourth axis)
Programming	HEIDENHAIN conversational programming
Program memory capacity	20 part programs 2 000 program blocks 1 000 program blocks in each program
Position data	Single-axis Cartesian coordinates; absolute or incremental
Unit of measurement	Millimeters or inches
Display step	Depending on encoders and machine parameters, e.g. 0.005 mm for graduation period of 20 µm
Input range	0.001 mm (0.000 5 in.) to 99 999.999 mm (3 937 in.); 0.001° to 99 999.999°
Max. range of traverse	+/– 10 000 mm
Maximum feed rate	30 000 mm/min
Maximum spindle speed	99 999 rpm
Number of tools in the tool table	99
Datum points	99
Data interface	RS-232-C/V.24
Data transfer rate	110, 150, 300, 600, 1 200 baud 2 400, 4 800, 9 600, 38 400 baud
Programming program section repeats	Subprograms; Program section repeats
Fixed cycles	Pecking; Tapping with a floating tap holder Circular hole pattern; Linear hole pattern; Rectangular pocket milling
Ambient temperature	Operation: 0° C to 45° C Storage: –30° C to 70° C
Weight	Approx. 6.5 kg
Power consumption	Approx. 27 W

Accessories

Electronic handwheels	
HR 130	For panel mounting
HR 410	Portable handwheel with permissive buttons

P

R

S

T

U

W

Z

Sequence of Program Steps
Milling an outside contour

Operating mode: PROGRAMMING AND EDITING

Program step			
1	**Open or select program**		
	Entries:	Program number Unit of measurement in the program	
2	**Calling tool data**		
	Entries:	Tool number Spindle axis	
	Separately:	Spindle speed	
3	**Tool change**		
	Entries:	Coordinates of the tool change position Radius compensation	
	Separately:	Feed rate (rapid traverse) and Miscellaneous function (tool change)	
4	**Approach starting position**		
	Entries:	Coordinates of the starting position Radius compensation (R0)	
	Separately:	Feed rate (rapid traverse) and Miscellaneous function (spindle ON, clockwise)	
5	**Move to (first) working depth**		
	Entries:	Coordinates of the (first) working depth Feed rate (rapid traverse)	
6	**Move to first contour point**		
	Entries:	Coordinates of the first contour point Radius compensation for machining	
	Separately:	Machining feed rate	
7	**Machining to last contour point**		
	Entries:	Enter all required data for all contour elements	
8	**Move to end position**		
	Entries:	Coordinates of the end position Radius compensation (R0)	
	Separately:	Miscellaneous function (spindle STOP)	
9	**Retract tool**		
	Entries:	Coordinates above the workpiece	
	Separately:	Feed rate (rapid traverse) and Miscellaneous function (end of program)	
10	**End of program**		

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH
Dr.-Johannes-Heidenhain-Straße 5
83301 Traunreut, Germany
☎ +49 (8669) 31-0
FAX +49 (8669) 5061
E-Mail: info@heidenhain.de

Technical support FAX +49 (8669) 31-1000
E-Mail: service@heidenhain.de
Measuring systems ☎ +49 (8669) 31-3104
E-Mail: service.ms-support@heidenhain.de
TNC support ☎ +49 (8669) 31-3101
E-Mail: service.nc-support@heidenhain.de
NC programming ☎ +49 (8669) 31-3103
E-Mail: service.nc-pgm@heidenhain.de
PLC programming ☎ +49 (8669) 31-3102
E-Mail: service.plc@heidenhain.de
Lathe controls ☎ +49 (711) 952803-0
E-Mail: service.hsf@heidenhain.de

www.heidenhain.de

Ve 00
284 679-24 · 7/2004 · pdf · Subject to change without notice