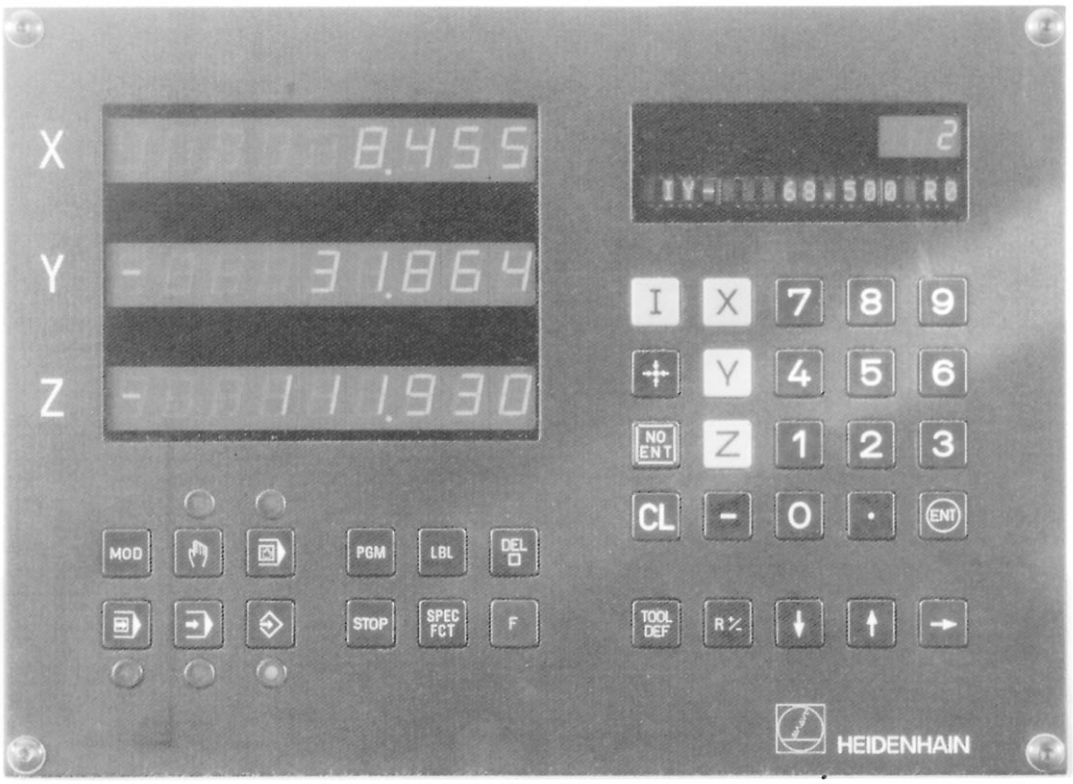


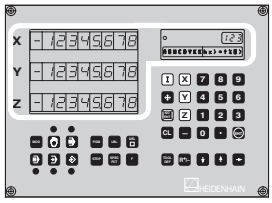
HEIDENHAIN

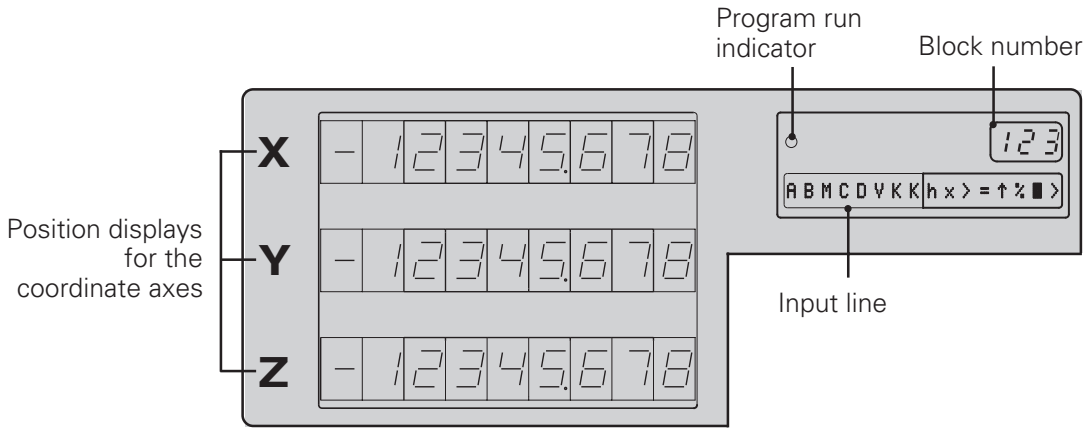
User's Manual

TNC 122

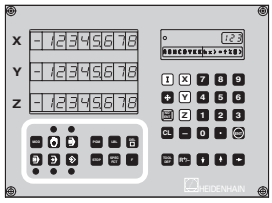
December 1994

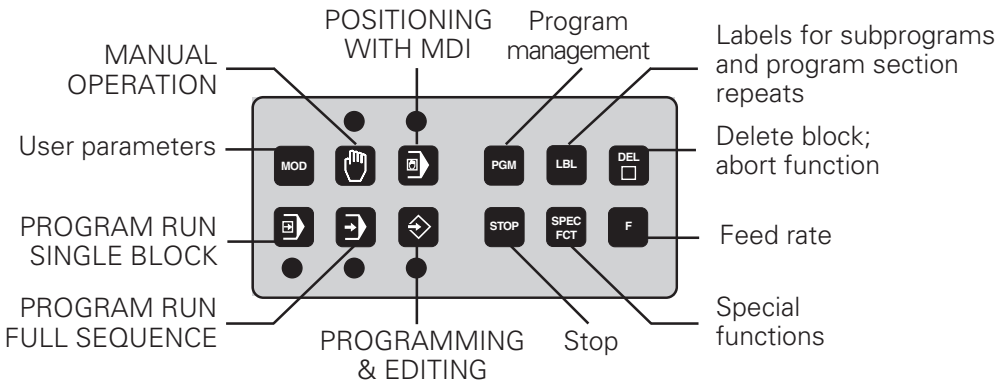
Display



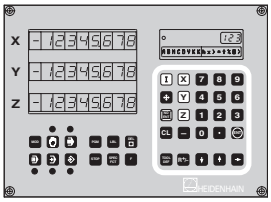


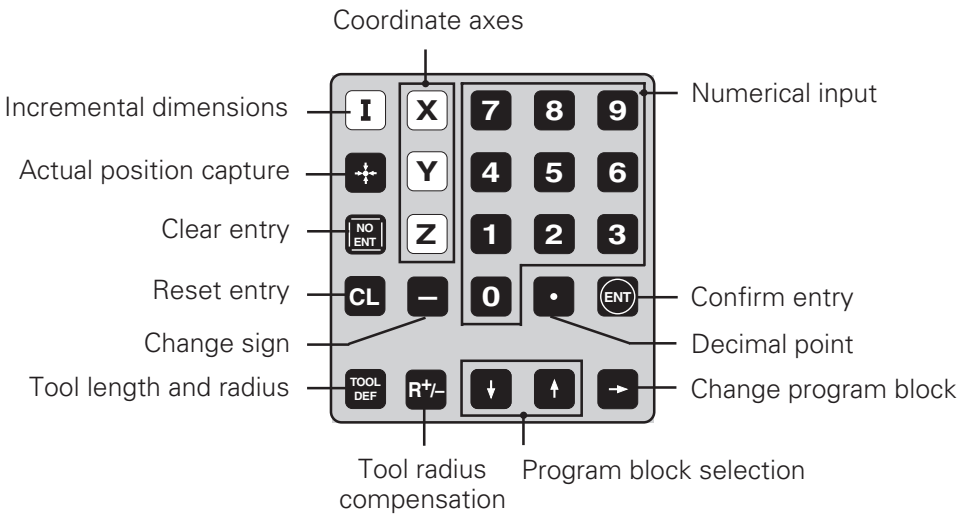
Operating modes, Programming





Input in all modes of operation





The TNC Guideline

From the workpiece drawing to program-controlled machining

Step	Task	TNC mode of operation	Starting on page
	Preparation		
1	Select tools	——	——
2	Set workpiece datum for coordinate system	——	——
3	Determine spindle speeds and feed rates	——	——
4	Switch on TNC and machine	——	15
5	Cross over reference marks		15
6	Clamp workpiece	——	——
7	Set datum/ set position displays		21
	Entering and testing part programs		
8	Enter part program or download over external data interface		from 31
9	Test run: Run program block by block without tool		51
10	If necessary: Optimize the part program		from 31
	Machining the workpiece		
12	Insert tool and run part program		51

TNC Accessory

Floppy disk unit

With the HEIDENHAIN FE 401 B floppy disk unit you can store programs from the TNC on diskette.

It is also a means of transferring programs created on a personal computer to the TNC.

The FE 401 B Floppy Disk Unit

Contents

Contents

Scope of this Manual

This manual describes the operation of the TNC 122 from the software version

Progr. 246 xxx **01**.

The three x's represent any numbers.

For detailed technical information, refer to the Technical Manual for the TNC 122.

TNC 122

TNC-Familie

What is NC?
NC stands for Numerical Control, 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, for "Touch Numerical Controls."

The **TNC 122** is a straight cut control for milling, drilling, and boring machines with up to three axes.

Differences from the TNC 121

The TNC 122 features the following improvements over the TNC 121:

- Larger program memory
- Tool compensation
- Programmable feed rate
- RS-232-C/V.24 data interface

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 or the feed rate

How to Use This Manual

As a **TNC beginner**, you can use the operating instructions as a step-by-step workbook. This part begins with a short introduction to some important basics concepts, and provides an overview of the available features. Then each feature is explained in detail, using a practical example that you can immediately try out on the machine — so you can't get lost in the theory. As a beginner you should work through all the examples presented.

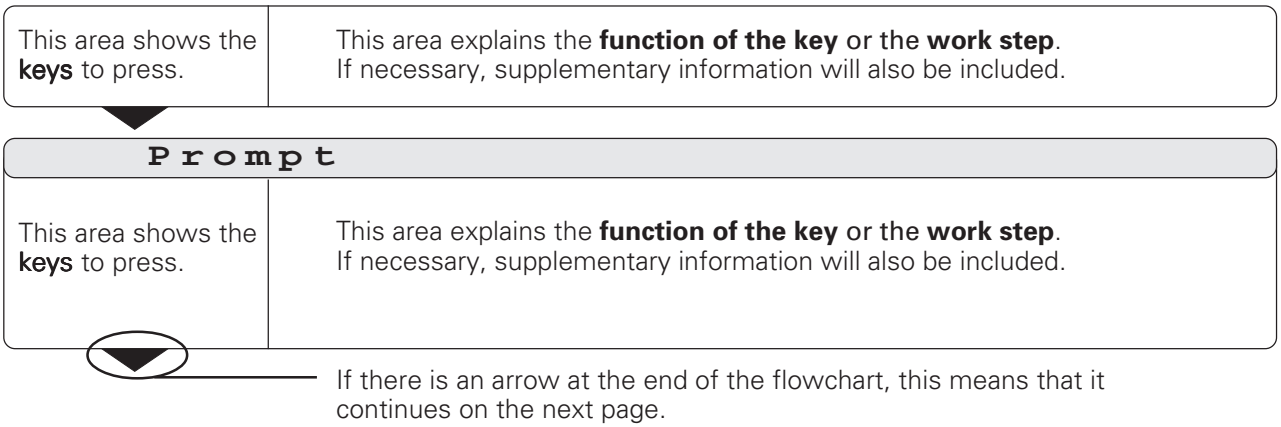
The **examples** are intentionally brief; it generally won't take you longer than 10 minutes to enter the example data.

As a **TNC expert**, you can use this manual 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.



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) above the input keypad.

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

Special Notes in This Manual

Especially important information is shown as a separate note in a gray box. Pay special attention to these notes. Ignoring them would prevent effective use of the control, or even result in damage to the tool or workpiece.

Symbols in the gray boxes

The symbols in the left of the gray boxes indicate the nature of the provided information.

General information
for example on the machine tool.function

Information for the machine tool builder
for example that he must implement a certain function

Essential information
for example that a certain tool is needed for the described function

NOTES

1
Fundamentals of Positioning

Reference system and coordinate axes

Reference system

In order to define positions on a surface one needs a reference system. 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." The network of horizontal and vertical lines on the globe constitute an absolute reference system in contrast to the "relative" definition of a position that is referenced to some other known location.

The illustration at right shows the 0° longitude at the Greenwich observatory and the 0° latitude at the equator.

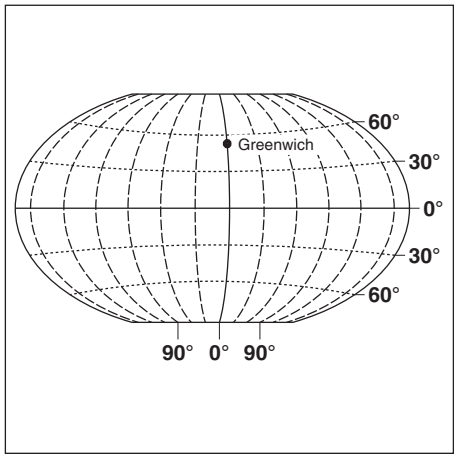


Fig. 1.1: The geographic coordinate system is an absolute reference system

Cartesian coordinate system

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

The figure at 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. X, Y and Z are the main axes of the Cartesian coordinate system.

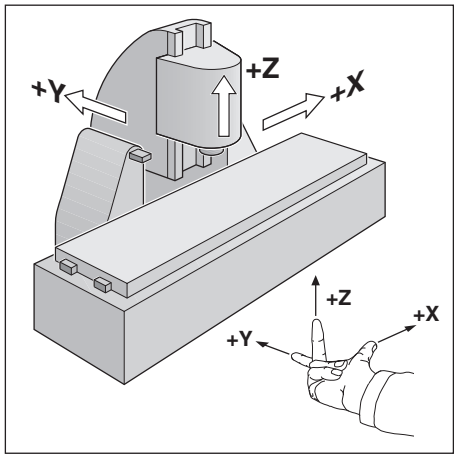


Fig. 1.2: Designations and directions of the axes on a milling machine

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).

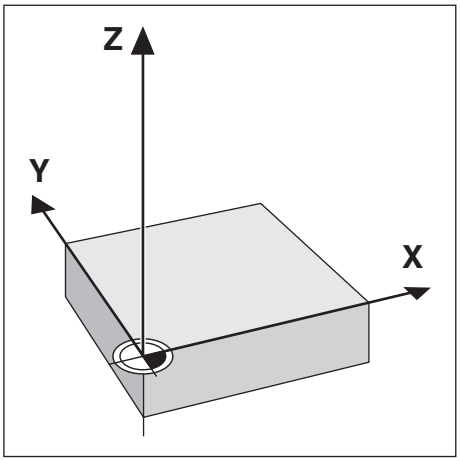


Fig. 1.3: The workpiece datum represents the origin of the Cartesian coordinate system

Example: Coordinates of hole ①:

X = 10 mm
Y = 5 mm
Z = 0 mm (hole depth: Z = – 5 mm)

The datum of the Cartesian coordinate system is located 10 mm from hole ① on the X axis and 5 mm from it in the Y axis (in negative direction).

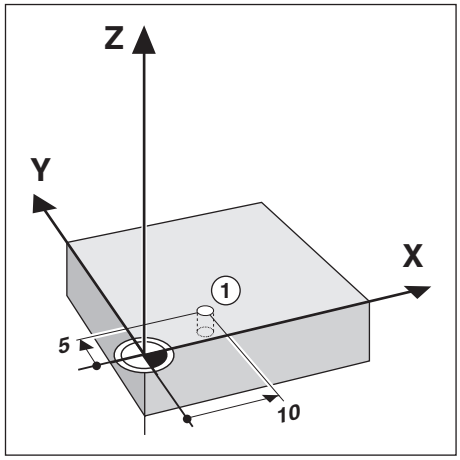


Fig. 1.4: Hole defines the coordinate system

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.

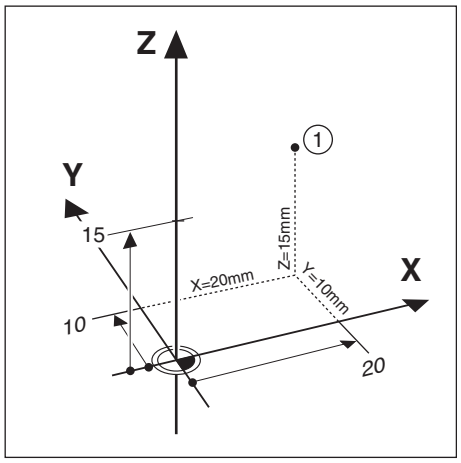


Fig. 1.5: Position definition through absolute coordinates

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.

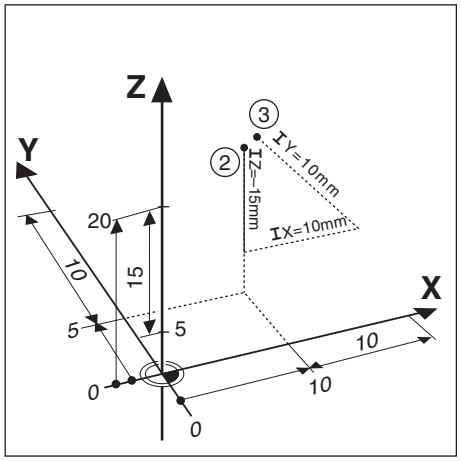


Fig. 1.6: Position definition through incremental coordinates

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.

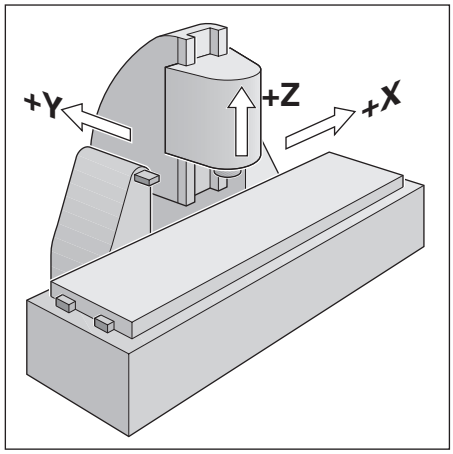


Fig. 1.7: On this machine the tool moves in the Y and Z axes; the workpiece moves in the X axis.

Position feedback

The position feedback encoders 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 with the aid of the encoders' reference marks.

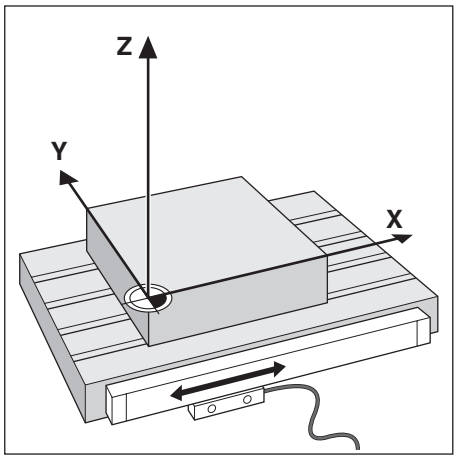


Fig. 1.8: Linear position encoder, here for the X axis

Reference marks

The scales of the position encoders have 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.

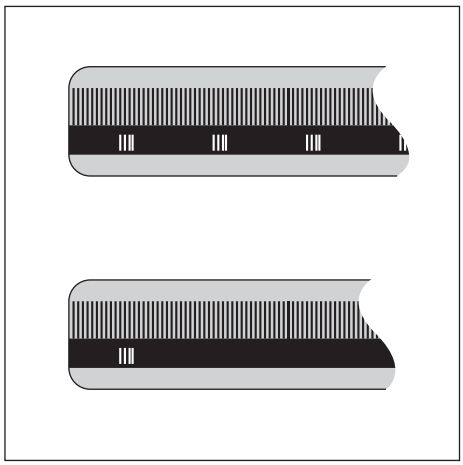


Fig. 1.9: Linear scales: above with distance-coded reference marks, below with one reference mark

2
Working with the TNC 122 – First Steps

Before you begin

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.

When you set a new datum point, the control automatically stores the new relationship between axis positions and display values.

Switch on the TNC

0 ⇒ 1	Switch on the TNC and the machine tool.
MEMORY TEST	
Please wait...	The TNC automatically checks its internal memory.
POWER INTERRUPTED	
CL	Clear the TNC message indicating that the power was interrupted.
NO CONTROL VOLTG	
I	Switch on the control voltage. The TNC automatically checks the function of the EMERGENCY STOP button.
REF TRAV ENT/NOE	
ENT	Select reference mark evaluation.
REF MARK XYZ	
Press and hold: X Y Z	Cross the reference marks in any direction: **Press and hold** 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 122 is now ready for operation in the MANUAL OPERATION mode.

If you do not wish to cross over the reference marks:

⇒ Answer the `REF TRAV ENT/NOE` dialog prompt with NO ENT (this feature must be implemented by the machine tool builder).

Operating modes

Selecting an operating mode makes a specific group of functions available.

Usable functions	Operating mode	Key
Moving the machine axes; Setting the datum	MANUAL OPERATION	
Entering positioning blocks and executing them block by block; Changing feed rate and miscellaneous functions; Entering tool data	POSITIONING WITH MANUAL DATA INPUT	
Storing working steps for small-lot production by • Keyboard entry • Teach-In Transferring programs through the data interface	PROGRAMMING AND EDITING	
Running programs blockwise	SINGLE BLOCK	
Running programs continuously	AUTOMATIC	

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

Error messages

If an error occurs while you are operating the TNC, a message will appear in plain language. You will find an overview of error message in Chapter 9.

To **clear** an error message:

⇒ Press the **CL** key.

Blinking error messages

W A R N I N G !

A blinking error messages means that the operational reliability of the TNC has been impaired.

If the TNC shows a blinking error message:

⇒ Write down the message.

⇒ Switch off the TNC and the machine tool.

⇒ Try to correct the error with the power off.

⇒ If the error cannot be corrected or if a blinking error message persists, call your service representative.

Selecting the position display mode

The TNC can show different types of position values for a tool position.

Fig 2.1 shows the following positions

- 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:

- Actual position ②
  The position at which the tool is presently located as referenced to the workpiece datum.
- Servo lag ③
  The difference between nominal ① and actual ② positions
- Actual position referenced to the scale reference point ④

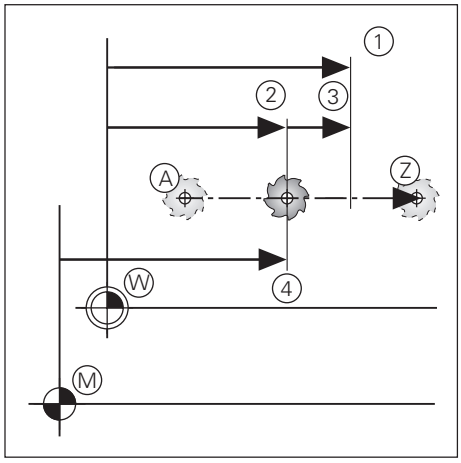


Fig 2.1: Tool and workpiece positions

To **change** position display modes:

⇒ Set another position display mode in the user parameter MP 7322 (see Chapter 8).

NOTES

3
Manual Operation and Setup

The TNC 122 provides two methods for manually moving the machine axes:

- Axis direction buttons
- Positioning with Manual Data Input (see Chapter 4)

Changing the feed rate F

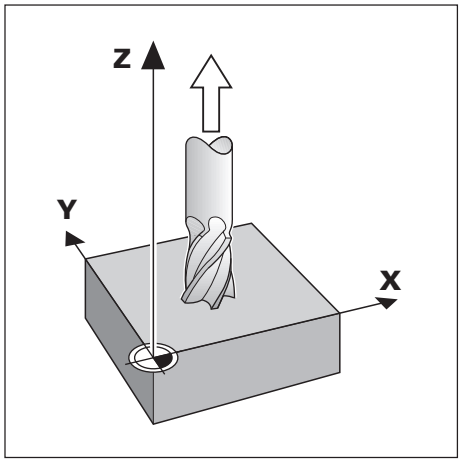
Some machines are equipped with a potentiometer to enable you to vary the feed rate.

Moving the machine axes with the axis direction buttons

In the MANUAL OPERATION mode you can move a machine axis by pressing the appropriate axis direction button on the machine control panel. As soon as you release the button the axis stops.

Continuing machine axis movement
With the user parameter MP7680 (see Chapter 8) you can set the TNC for continuing machine axis movement. The machine then continues to move the axis after you have released the axis direction button. To stop the machine axis you must press a button again (see example 2 below).

Example: Moving the machine axis with the machine axis direction button in the Z+ direction (retracting the tool)



Example 1: Moving the machine axis

Mode of operation: MANUAL OPERATION

Press and hold: Z	Press the direction button, e.g. Z, and hold it as long as you wish the machine axis to move.

Example 2: Moving the machine axis (continuing movement)

Mode of operation: MANUAL OPERATION

Together: Z I	To start the axis, press an axis direction button, such as Z, and the NC start button at the same time.
0	Stop the axis with the NC stop button.

Entering tool length and radius

You can enter the length and radius of you tool in the TNC. The TNC includes the tool radius in the position value when you position with radius compensation (see p. 21).

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

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

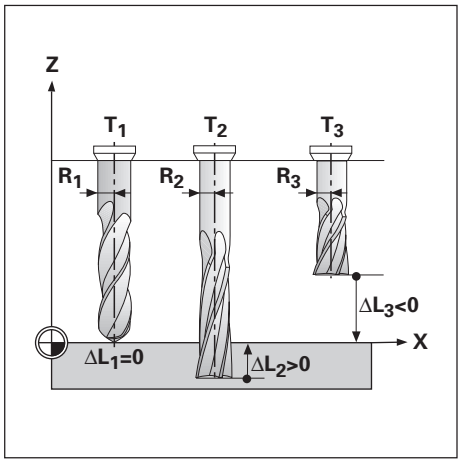


Fig. 3.1: Tool lengths and radii

Position display in the tool axis

User parameter MP7285 defines whether the tool axis display value shows the position of the tool tip or the tool datum.

Checking the tool data

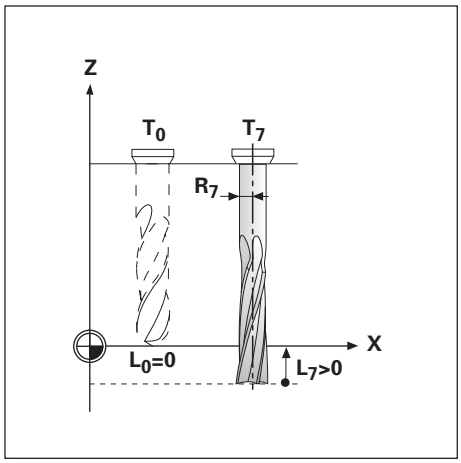
To display the tool data:
⇒ Press the TOOL DEF key.

To display the tool length and tool axis:
⇒ Press the downward-arrow key twice.

To return to the position display:
⇒ Press the NO ENT key.

Example: Entering the tool length and radius

Tool radius:	8 mm
Tool length:	12 mm
Tool axis:	Z

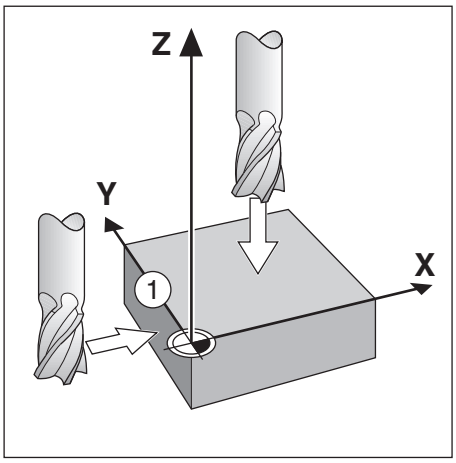


TOOL DEF	Call the tool definition function.
RADIUS = ...	
8 ENT	Enter the tool RADIUS (8 mm). Confirm your entry.
LENGTH = ...	
1 2 ENT	Enter the tool LENGTH (12 mm). Confirm your entry.
AXIS = .	
Z ENT	Enter the tool AXIS (Z). Confirm your entry.

Setting the datum: Moving to the datum surface and entering the actual value

To set the datum, you move the tool to the respective datum surfaces and enter the tool position as datum.

Example: Setting the datum in the X and Z axes



Working plane: X / Y

Tool axis: Z

Tool radius: R = 5 mm

Sequence for datum setting in this example: X, Z

Procedure

⇒ Insert the tool.
⇒ Enter the tool data.
⇒ Switch on the spindle, e.g. with the miscellaneous function M 3.

Mode of operation: MANUAL OPERATION

	Touch surface ① with the tool.
X	Select the X axis.
DATUM X = X =	
– 5 ENT	Enter the position of the tool center (X = – 5 mm) and transfer the X coordinate of the datum.
	Touch the top surface with the tool.
Z	Select the Z axis.
DATUM Z = Z =	
0 ENT	Enter the position of the tool tip (Z = 0 mm) and transfer the Z coordinate to the display.

NOTES

4
Positioning with Manual Data Input (MDI)

For many simple machining tasks, for example if a part is to be machined 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 execute the working steps as you enter them 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.

Hole patterns

he POSITIONING WITH MDI mode of operation also supports the TNC "Cycles" (see Chapter 5):

- Bolt hole circle patterns
- Linear hole patterns

Before you machine the part

⇒ Insert the tool.
⇒ Pre-position the tool so that the tool and workpiece will not be damaged during workpiece approach.
⇒ Select an appropriate feed rate F.
⇒ Select an appropriate spindle speed S.
⇒ Switch on the spindle, e.g. with the miscellaneous function M3.

Taking the tool radius into account

The TNC can compensate the tool radius (see Fig. 4.1). This allows you to enter workpiece dimensions directly from the drawing. The TNC automatically lengthens (R+) or shortens (R–) traverse by the tool radius.

Entering tool data

⇒ Press the TOOL DEF key
⇒ Enter in sequence the tool radius, length, and axis.

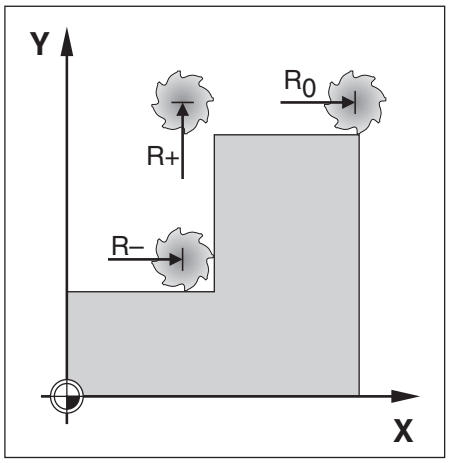


Fig. 4.1: Tool radius compensation

Entering and changing the feed rate F

Example: Enter the feed rate F

F	Select the feed rate function F for the next tool movement.
F 1 0 0	F blinks: Enter the feed rate F, e.g. 100 mm/min.
ENT	Confirm feed rate F for the next tool movement.

Changing the feed rate F

Some machines are equipped with a potentiometer to allow you to vary the feed rate.

Entering the miscellaneous function M

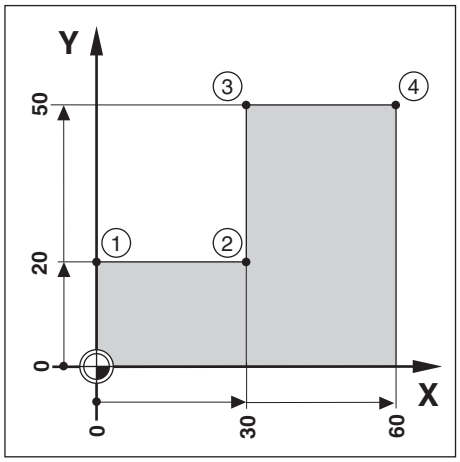
The machine tool builder determines which miscellaneous functions are available on your TNC and what effect they have.

SPEC FCT	Press the SPEC FCT key for special functions.
Repeatedly ↓	Page to M FUNCTION.
M FUNCTION ENT	Select M FUNCTION.
M 0 3 ENT	M blinks: Enter the desired M function, e.g. M3 (spindle on, clockwise). Confirm your entry.
I	Execute the M function.

Entering and moving to positions

For simple tasks, use the POSITIONING WITH MDI mode of operation to machine the dimensions as you enter them.

Example: Milling a shoulder



The coordinates are entered as absolute dimensions referenced to the workpiece datum.

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

Procedure:

⇒ Enter the tool data.
⇒ Move the tool to a good starting position (e.g. X = Y = – 20 mm).
⇒ Move the tool to the milling depth.

Mode of operation: POSITIONING WITH MDI

Y	Select the Y axis.
Y R 0 2 0 R+/– ENT	Y blinks: Enter the nominal position for corner point ①: Y = + 20 mm, select the tool radius compensation: R + and confirm your entry.
I	Move the tool to the programmed position.
X	Select the X axis.
X R 0 3 0 R+/– ENT	X blinks: Enter the nominal position for corner point ②: Y = + 30 mm, select the tool radius compensation: R – and confirm your entry.
I	Move the tool to the programmed position.

Y	Select the Y axis.
Y R 0	
5 0 R+/- ENT	Y blinks: Enter the nominal position for corner point ③ : Y = +50 mm, Select tool radius compensation: R + and confirm your entry.
I	Move the tool to the entered position.
X	Select the X axis.
X R 0	
6 0 R+/- ENT	X blinks: Enter the nominal position for corner point ④ : Y = +60 mm, Select tool radius compensation: R + and confirm your entry.
I	Move the tool to the entered position.

Hole patterns

The hole pattern functions **BOLT HOLE CIRCLE** and **LINEAR PATTERN** are provided in the POSITIONING WITH MDI mode of operation.

You select the BOLT HOLE CIRCLE or LINEAR HOLE PATTERN function and enter the necessary data. This data, such as the number of holes and the coordinates of the first hole, is normally shown on the production drawing.

The **TNC** calculates the positions of the holes.

Pre-positioning the drill

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

Hole depth

If you wish to use a spindle sleeve to drill the holes manually:

⇒ Answer the dialog prompt `DEPTH =` with NO ENT.

Input for a bolt hole circle

- Full circle or circle segment
- Number of holes
- Center point coordinates and radius of the circle
- Starting angle (angular position of first hole)
- For circle segment: angle step between the holes
- Hole depth

Input for linear hole patterns

- 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
- Hole depth

Drilling the hole pattern

After you have entered all the data:

⇒ Press the NC start key repeatedly.

The TNC moves the axes one at a time in the working plane and the tool axis. After drilling it returns the tool to the starting height.

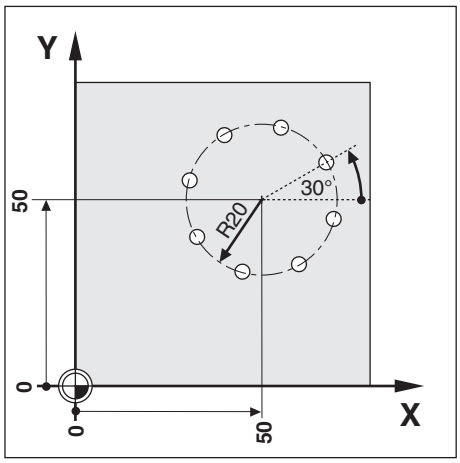
Skipping holes

If you wish to skip certain holes, or bore in a different sequence than that calculated by the TNC:

⇒ Select the desired hole with the upward and downward arrow keys.

Example: Entering and machining a bolt hole circle

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



Mode of operation: POSITIONING WITH MDI

SPEC FCT	Press the SPEC FCT key for special functions.
Repeatedly ↓	Page to the BOLT HOLE CIRCLE function.
B O L T H O L E C I R C L E	
ENT	Select the BOLT HOLE CIRCLE function.
Repeatedly ↓	Page to the FULL CIRCLE function.
F U L L C I R C L E	
ENT	Select FULL CIRCLE
N O H L =	
8 ↓	Enter the number of holes NO HL (8). Confirm your entry and continue the dialog.
C E N T X =	
5 0 ↓	Enter the X coordinate of the bolt hole circle center (X = 50 mm). Confirm your entry and continue the dialog.
C E N T Y =	
5 0 ↓	Enter the Y coordinate of the bolt hole circle center (Y = 50 mm). Confirm your entry and continue the dialog.
R A D I U S =	
2 0 ↓	Enter the RADIUS of the bolt hole circle (20 mm). Confirm your entry and continue the dialog.
A N G L E =	
3 0 ↓	Enter the starting ANGLE from the X axis to the first hole (30°). Confirm your entry and continue the dialog.

D E P T H =	
8 ↓	Enter the hole DEPTH (8 mm). Confirm your entry and continue the dialog.

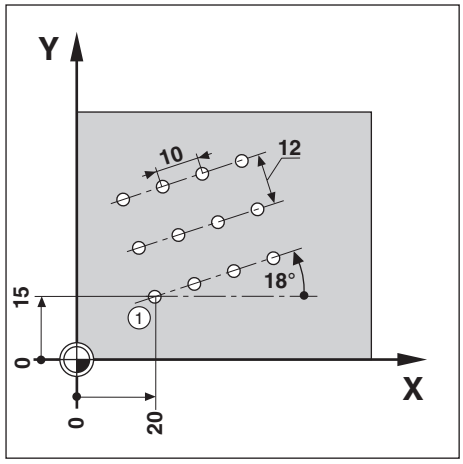
B O L T H O L E C I R C L E ?	
ENT	Start the bolt hole circle.

C Y C L F U L L C I R C L E	
I	Start the FULL CIRCLE cycle.

B L T C I R H O L E 1 ...	
Repeatedly I	For each hole move the axes in the working plane and drill until all holes in the full circle are completed.

Example: Entering and machining linear hole patterns

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
Hole depth:	8 mm

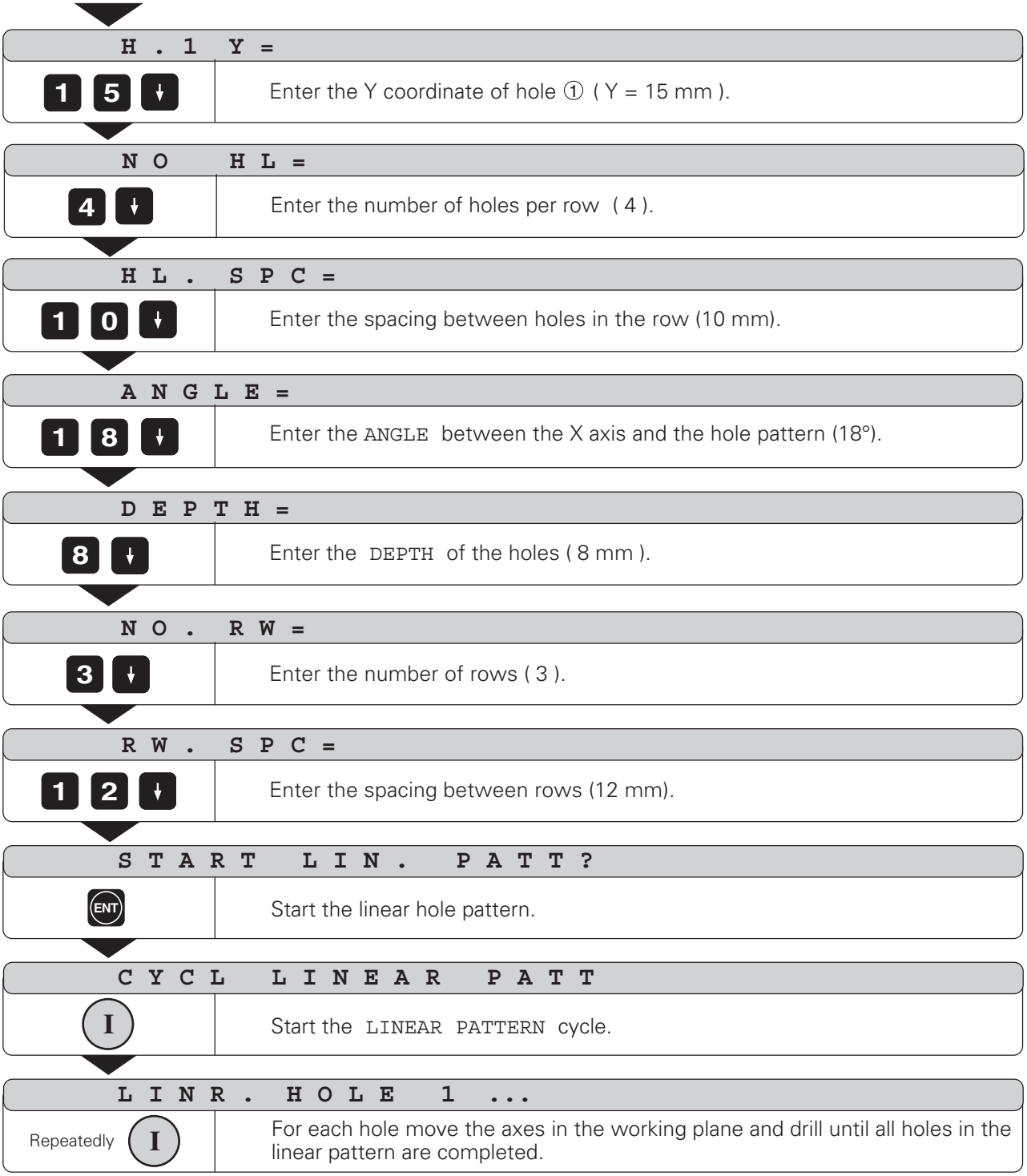


Mode of operation: POSITIONING WITH MDI

SPEC FCT	Press the SPEC FCT key for special functions.
Repeatedly ↓	Page to the LINEAR PATTERN function.

L I N E A R P A T T E R N	
ENT	Select the LINEAR PATTERN function.

H . 1 X =	
2 0 ↓	Enter the X coordinate of hole ① (X = 20 mm). Confirm your entry and continue the dialog.


H . 1 Y =
1 5
Enter the Y coordinate of hole ① (Y = 15 mm).
N O H L =
4
Enter the number of holes per row (4).
H L . S P C =
1 0
Enter the spacing between holes in the row (10 mm).
A N G L E =
1 8
Enter the ANGLE between the X axis and the hole pattern (18°).
D E P T H =
8
Enter the DEPTH of the holes (8 mm).
N O . R W =
3
Enter the number of rows (3).
R W . S P C =
1 2
Enter the spacing between rows (12 mm).
S T A R T L I N . P A T T ?
ENT
Start the linear hole pattern.
C Y C L L I N E A R P A T T
I
Start the LINEAR PATTERN cycle.
L I N R . H O L E 1 ...
Repeatedly
I
For each hole move the axes in the working plane and drill until all holes in the linear pattern are completed.

5 Programming

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

Programs in the TNC

The part programs stored in the TNC contain the working steps for machining a part. You can edit, add to and run these programs as often as you wish.

You can store programs on floppy disk 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.

Program storage capacity
The TNC 122 stores up to 20 programs with a maximum of 500 NC blocks. A single program can contain up to 500 NC blocks.

Programmable functions

- Interrupt the program (STOP)
- Feed rate F
- Miscellaneous function M
- Nominal position values
- Teach-In: capturing the actual position
- 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 places in the program.

Tool and workpiece movement

During workpiece machining, the machine moves an axis by moving either the tool or 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 the tool

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

What happens with the completed programs?

The completed program is used to machine the part in the PROGRAM RUN mode of operation. See Chapter 7 for an explanation of this mode.

Entering the program number

Select a program with a number from 1 to 20.

Mode of operation: PROGRAMMING AND EDITING

PGM	Select program management.
ENT	Switch to the EDIT PGM function.
E D I T P G M	
ENT	Select the EDIT PGM function.
P G M N O . =	
7 ENT	The equal sign blinks: Enter the PGM-NO., for example 7. Confirm your entry. Now you can enter and edit the program.

The BEGIN block of the selected program appears.

Deleting programs

If you no longer need certain programs, or if you need to make space in the TNC's memory, you can **delete** programs.

Mode of operation: PROGRAMMING AND EDITING

PGM	Select program management.
↓	Switch to the DELETE PGM function.
D E L E T E P G M	
ENT	Select the DELETE PGM function.
P G M N O . =	
5 ENT	The equals sign blinks: enter the PGM NO. 5 to erase program 5.

After deletion the BEGIN block of the deleted program appears. The contents of the program is deleted and the BEGIN and END blocks of the deleted program remain in the TNC's memory.

Erase all programs

⇒ Use the upward arrow key to go from the DELETE PGM function to the DELETE ALL PGM function.
⇒ Press ENT to erase all programs.

Selecting program blocks

Current block

The current block appears in the entry line above the numeric keypad. The block number appears to the right and above the entry line.

The TNC inserts new blocks behind the current block. No more blocks can be entered if the END PGM block appears in the entry line

Overview of functions

Function	Key
Select the next block	↓
Select the previous block	↑

Go directly to a program block

In large programs it can take a long time to scroll to the desired block using the arrow keys. A quicker way is to use the GOTO function to go directly to the desired block.

⇒ Enter the number of the desired block.

⇒ Confirm your entry with the ENT key.
The desired block appears in the entry line.

Changing program blocks

You can make changes in program blocks the incorrect numerical entries in a program.

Clearing incorrect numerical entries

If you notice an incorrect numerical entry immediately after you've made it, you can clear it and try again:

⇒ Press the CL key.

Confirming a change

Any change made with CL **must** be confirmed with ENT to become effective!

Example: Changing a program block

Mode of operation: PROGRAMMING AND EDITING

↑ / ↓	Move to the program block that you wish to change.
→	Select the block for editing. The display (e.g. the axis designation) starts blinking.
2 0	Enter the desired change, for example a new nominal position value (20).
ENT	Confirm the change.

Deleting program blocks

You can delete any blocks in an existing program except the BEGIN and END blocks

To delete a block:

⇒ Use the arrow keys to move to the block, or enter the block number.

⇒ Press the DEL □ key.

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

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

⇒ Select the last block of the program section.

⇒ Press the DEL □ key repeatedly until all the blocks in the section have been deleted.

Feed rate F and miscellaneous function M

The feed rate F and miscellaneous function M are entered as separate blocks. They become effective as soon as the TNC has run the block in which they are programmed.

These blocks must be run **before** the positioning blocks for which they are intended.

Entering the feed rate F

The machining feed rate is "modal." That means that the entered feed rate remains effective until you replaced it by entering a new one.

Example

Mode of operation: PROGRAMMING AND EDITING

F	Press the F key for feed rate. A blinking F appears.
F E E D R A T E F	
1 0 0	Enter the desired feed rate F, for example 100 mm/min.
ENT	Confirm the feed rate F for the following positioning blocks.

Varying the feed rate

Some machines are equipped with a potentiometer to enable you to vary the feed rate.

Entering the miscellaneous function M

The machine tool builder determines which miscellaneous functions are available on your TNC and what effect they have.

Example: Entering a miscellaneous function

Mode of operation: PROGRAMMING AND EDITING

SPEC FCT	Press the SPEC FCT key for special functions.
Repeatedly ↓	Page to `M FUNCTION`.
M FUNKTION	
ENT	Select `M FUNCTION`.
M	
3	Enter the miscellaneous function e.g. M3 (Spindle ON, clockwise).
ENT	Confirm your `M function` entry.

Entering a program interruption

You can divide your program into logical sections by setting stop blocks. The TNC interrupts the program at the stop block and resumes it when you press a button.

Mode of operation: PROGRAMMING AND EDITING

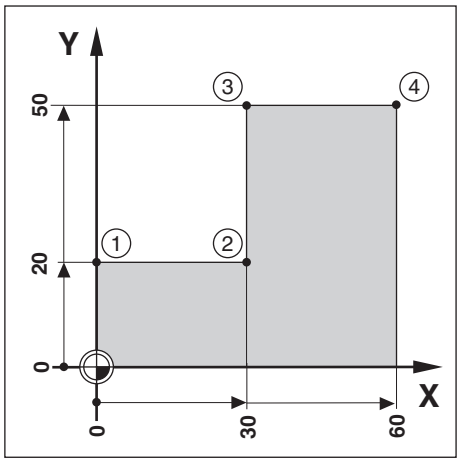
STOP	Enter the `STOP` block in the program.

To restart a program after an interruption

⇒ Press the NC-Start button

Entering workpiece positions

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 programming steps

⇒ Press the PGM key.
⇒ Key in the number of the program you want to work on, and press ENT.
⇒ Enter the nominal positions.

Running a completed program

Once a program has been completed it can be executed in the PROGRAM RUN mode (see Chapter 10).

Example: Entering a nominal position in a program (Block 9 in the example)

X	Select the coordinate axis (X axis).
X R 0	
3 0 R+/-	Enter the nominal position value, for example 30 mm and select tool radius compensation R – .
ENT	Confirm your entry. The nominal position now appears in the program block display.

Program blocks

0	BEGIN PGM 10		Start of program, program number
1	F 9999		High feed rate for pre-positioning
2	Z+20.000		Clearance height
3	X–20.000	R0	Pre-position the tool in the X axis
4	Y–20.000	R0	Pre-position the tool in the Y axis
5	Z–10.000		Move tool to milling depth
6	F 200		Machining feed rate
7	M 3		Spindle ON, clockwise
8	Y+20.000	R+	Y coordinate, corner ①
9	**X+30.000**	**R–**	X coordinate, corner ②
10	Y+50.000	R+	Y coordinate, corner ③
11	X+60.000	R+	X coordinate, corner ④
12	F 9999		High feed rate for retracting
13	Z+20.000		Clearance height
14	M 2		Stop program run, spindle OFF, coolant OFF
15	END PGM 10		End of program, program number

Actual-position capture: Teach-In programming

With teach-in programming, you enter the position values by moving to the position and then transferring the actual position value into the program.

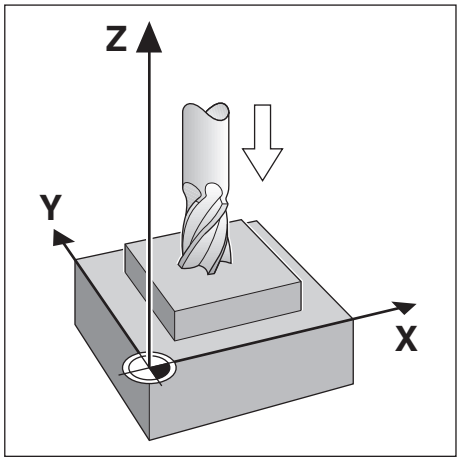
Changing the captured position values

Teach-in blocks can be edited later just like any other program blocks.

Selecting radius compensation

If you wish to change the radius compensation:
⇒ Press the R +/– key.

Programming example: Capturing a Z-coordinate value (top surface of workpiece) for a part program



Mode of operation: PROGRAMMING AND EDITING

	Move the tool until it touches the surface of the workpiece.
Z	Select the axis, for example Z.
Z	
	Capture the position of the tool point for the program.
ENT	Store the position in the tool axis (Z).

Hole patterns in programs

The BOLT HOLE CIRCLE and LINEAR PATTERN cycles can also be entered in a part program and saved for repeated execution. Each item of information then comprises its own program block.

These blocks are introduced by a block with a block number, followed by the word CYCL and the name of the cycle. The cycles contain all information required by the TNC for machining a hole pattern.

The TNC executes a hole pattern automatically as soon as it reaches the cycle in the program.

Cycles must be complete

Do not delete any blocks from the cycle. If you do, it will provoke the error message CYCLE INCOMPLETE when the program is executed.

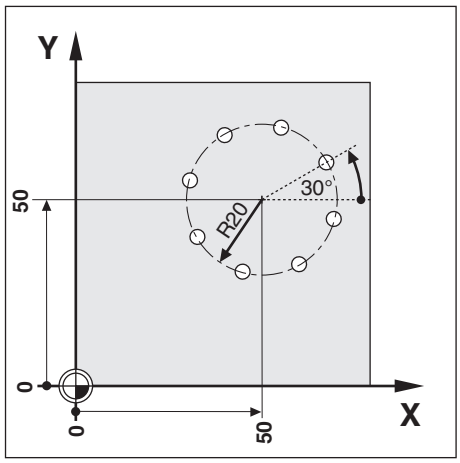
Entering cycles

Press the SPEC FCT key and select the desired cycle. The TNC automatically asks for all data required to execute the cycle.

Bolt hole circle

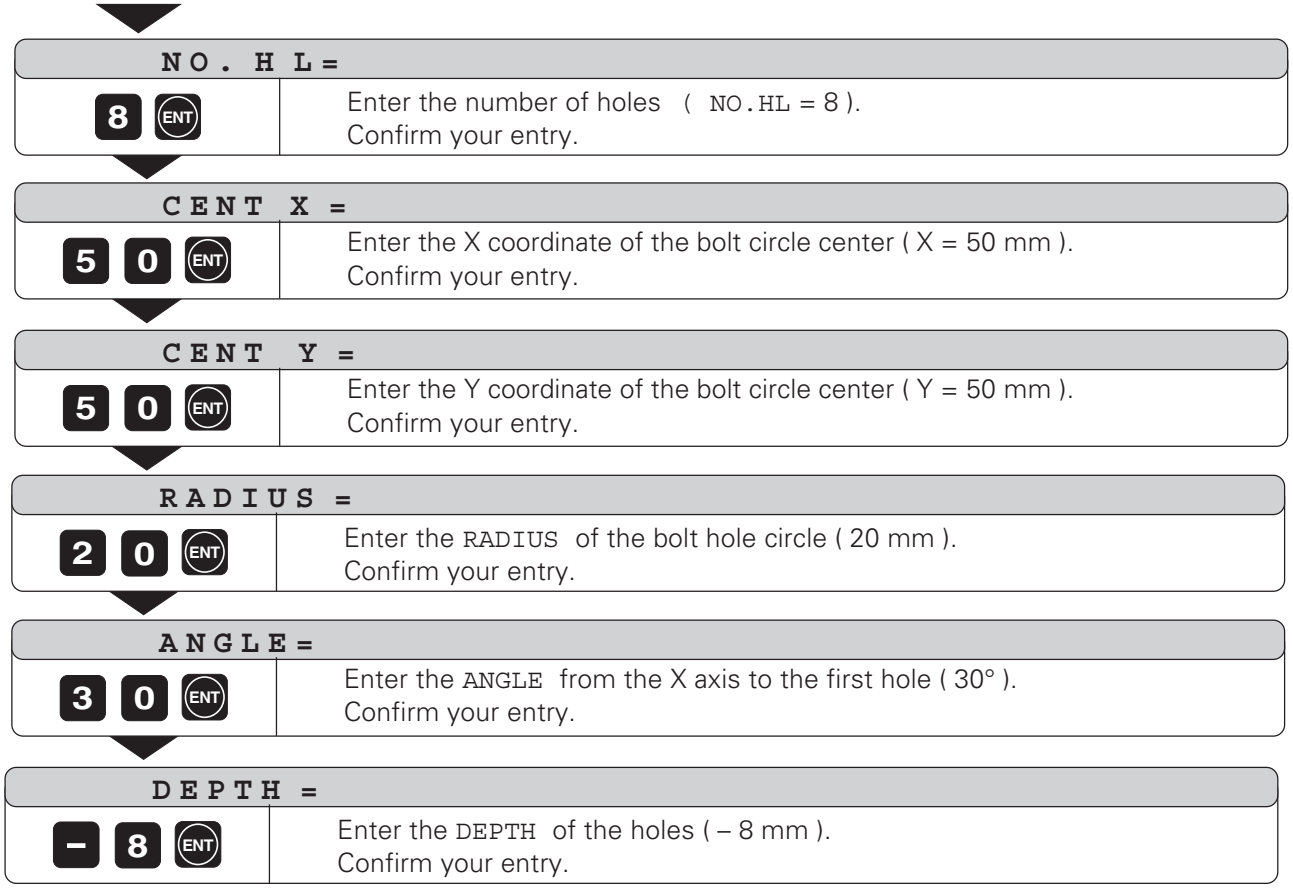
Programming example: FULL CIRCLE cycle

Number of holes NO.HL :	8
Center point coordinates:	CENT X = 50 mm
	CENT Y = 50 mm
Bolt hole circle radius RADIUS :	20 mm
Starting angle between X axis and first hole ANGLE :	30°
Drilling depth DEPTH :	– 8 mm



Mode of operation: PROGRAMMING AND EDITING

SPEC FCT	Press the SPEC FCT key for special functions.
Repeatedly ↓	Page to the BOLT HOLE CIRCLE function.
B O L T H O L E C I R C L E	
ENT	Select the BOLT HOLE CIRCLE function.
F U L L C I R C L E	
ENT	Select FULL CIRCLE



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+20.000	Clearance height
3	M 3	Spindle ON, clockwise
4	**CYCL FULL CIRCLE**	The data for the FULL CIRCLE cycle follow this block
5	**NO.HL = 8**	Number of holes
6	**CENT X= 50.000**	X coordinate of the center of the bolt circle
7	**CENT Y= 50.000**	Y coordinate of the center of the bolt circle
8	**RADIUS= 20.000**	Radius
9	**ANGLE= 30.000**	Starting angle of first hole
10	**DEPTH= - 8.000**	Depth of holes
11	M 2	Stop program run, spindle STOP, coolant OFF
12	END PGM 40 MM	End of program, program number, unit of measurement

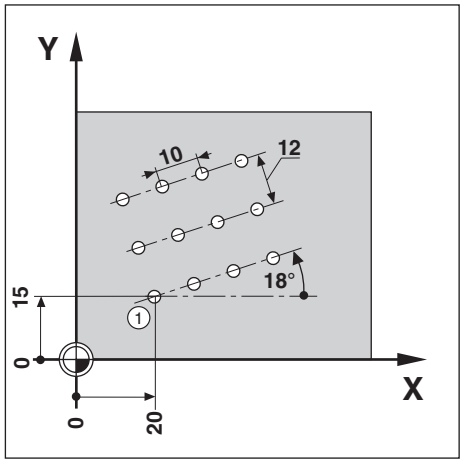
For a **circle segment** (CYCL CIRCL SEGMT) you **also** enter the angle step (ANGLE) between the holes (after the starting angle).

The bolt hole circle is then executed in the PROGRAM RUN mode of operation (see Chapter 7).

Linear hole patterns

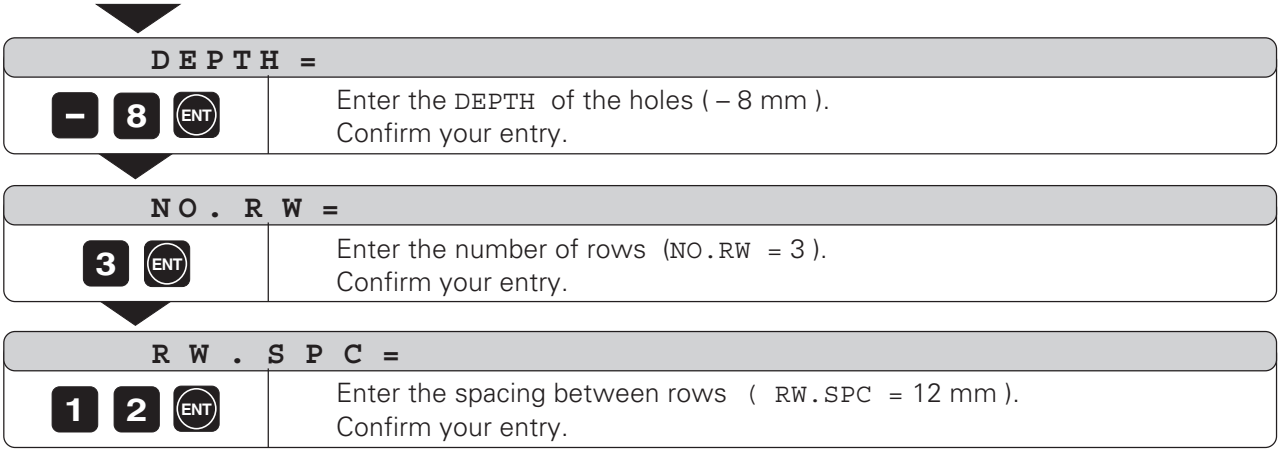
Programming example: `LINEAR PATTERN` cycle

X coordinate of the first hole ①	`H.1 X` = 20 mm
Y coordinate of the first hole ①	`H.1 Y` = 15 mm
Number of holes per row `NO.HL`	4
Hole spacing `HL.SPC`	10 mm
Angle between hole row and X axis `ANGLE`	18°
`DEPTH` of holes	– 8 mm
Number of rows `NO.RW`	3
Row spacing `RW.SPC`	12 mm



Mode of operation: PROGRAMMING AND EDITING

SPEC FCT	Press the SPEC FCT key for special functions.
B O L T H O L E C I R C L E ?	
↓ ENT	Select `BOLT HOLE CIRCLE`.
H . 1 X =	
2 0 ENT	Enter the X coordinate of hole ① (X = 20 mm). Confirm your entry.
H . 1 Y =	
1 5 ENT	Enter the Y coordinate of hole ① (Y = 15 mm). Confirm your entry.
N O . H L =	
4 ENT	Enter the number of holes per row (`NO.HL` = 4). Confirm your entry.
H L . S P C =	
1 0 ENT	Enter the hole spacing in the row (`HL.SPC = 10` mm). Confirm your entry.
A N G L E =	
1 8 ENT	Enter the `ANGLE` between the X axis and the rows of holes (`ANGLE = 18°`). Confirm your entry.



Program blocks

0	BEGIN PGM 50	Start of program, program number
1	F 9999	High feed rate for pre-positioning
2	Z+20.000	Clearance height
3	M 3	Spindle ON, clockwise
4	**CYCL LINEAR PATT**	The data for the LINEAR PATTERN cycle follow this block
5	**H.1 X= 20.000**	X coordinate of first hole
6	**H.1 Y= 15.000**	Y coordinate of first hole
7	**NO.HL= 4**	Number of holes per row
8	**HL.SPC= 10.000**	Distance between holes on the row
9	**ANGLE= 18.000**	Angle between the rows and the X axis
10	**DEPTH= –8.000**	Depth of the holes
11	**NO.RW= 3**	Number of rows
12	**RW.SPC= 12.000**	Spacing between rows
13	M 2	Stop program run, spindle STOP, coolant OFF
14	END PGM 50	End of program, program number

The hole pattern is then executed in the PROGRAM RUN mode of operation (see Chapter 7).

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.

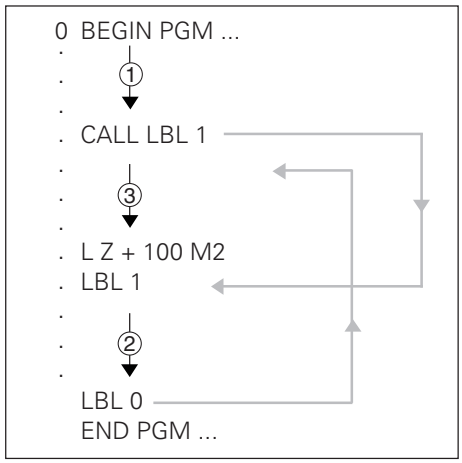


Fig 8.1: Operating sequence of a subprogram

Inserting program markers (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 that is to be repeated.

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

Label call
Subprogram and program sections are called within the program with a CALL L command.

The command **CALL L 0** is **not allowed**.

Subprograms:
When it reaches a CALL L block, the TNC immediately executes the called subprogram.

Program section repeats:
The TNC repeats the program section above the CALL L block. Together with the CALL L command you also enter the number of desired repetitions.

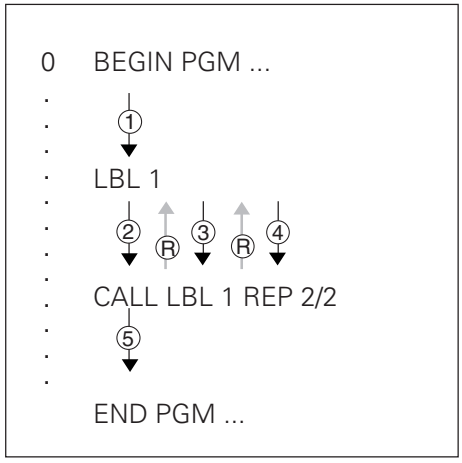


Fig. 8.2: Operating sequence of a program section repeat

Nesting

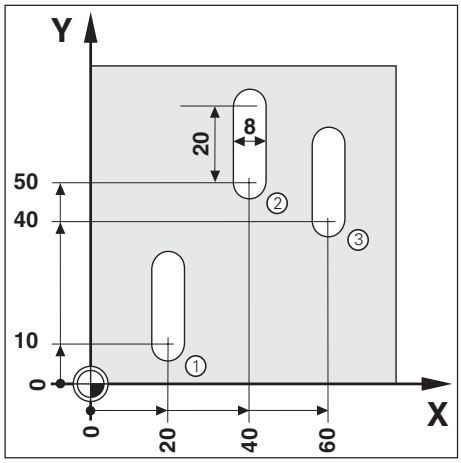
You can run subprograms and repeat program sections within other subprograms and program sections. This is called nesting. An example of nesting is when you call a subprogram from within another subprogram.

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 a label for a subprogram

Mode of operation: PROGRAMMING AND EDITING

LBL	Select the LBL function.
S E T = E N T / C A L L = L B L	
ENT	Select `SET` to set a label.
L B L ...	
5 ENT	Enter a label number. Confirm your entry. The resulting program block is: `LBL 5`

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 L`

Mode of operation: PROGRAMMING AND EDITING

LBL	Select the LBL function.
S E T = E N T / C A L L = L B L	
LBL	Select `CALL` to call a label.
C A L L L 0	
5 ENT	Enter the label number of the desired subprogram. Confirm your entry.
C A L L L 5 R	
NO ENT	R stands for "repetitions" and has no significance for subprogramming. Skip this prompt by pressing NO ENT. The resulting program block is: `CALL L5`

After a CALL L 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.

Program blocks

0	BEGIN PGM 60		Start of program, program number
1	F 9999		High feed rate for pre-positioning
2	Z+20.000		Clearance height
3	X+20.000	R0	X coordinate infeed point slot ①
4	Y+10.000	R0	Y coordinate infeed point slot ①
5	M 3		Spindle ON, clockwise
6	CALL L 1		Call subprogram 1: execute blocks 16 to 20
7	X+40.000	R0	X coordinate infeed point slot ②
8	Y+50.000	R0	Y coordinate infeed point slot ②
9	CALL L 1		Call subprogram 1: execute blocks 16 to 20
10	X+60.000	R0	X coordinate infeed point slot ③
11	Y+40.000	R0	Y coordinate infeed point slot ③
12	CALL L 1		Call subprogram 1: execute blocks 16 to 20
13	Z+20.000		Clearance height
14	M 2		Stop program run, spindle STOP, coolant OFF
15	**LBL 1**		Start of subprogram 1
16	**F 200**		Machining feed rate during subprogram
17	**Z–10.000**		Infeed to slot depth
18	**IY+20.000**	**R0**	Mill slot
19	**F 9999**		High feed rate for retracting and pre-positioning
20	**Z+2.000**		Retract
21	**LBL 0**		End of subprogram 1
22	END PGM 60		End of program, program number

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.

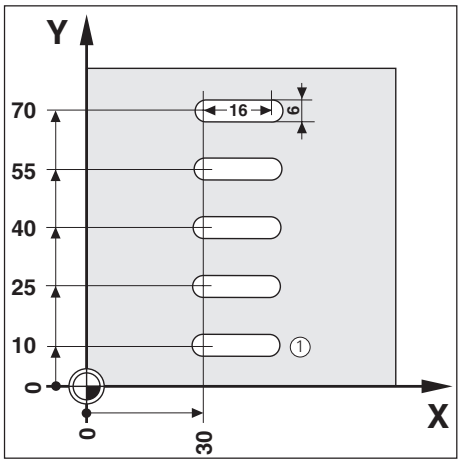
`CALL LBL` block for a program section repeat

Example of a call label block: `CALL L 1  R10 / 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 repetitions you programmed. The number **behind** the slash is the number of repetitions remaining to be run.

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

Mode of operation: PROGRAMMING AND EDITING

LBL	Select the LBL function.
`S E T = E N T / C A L L = L B L`	
ENT	Select `SET` to set a label.
`L B L ...`	
5 ENT	Enter the label number. Confirm your entry. The resulting program block is `LBL 5`

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

Example: Entering a program section repeat: CALL L

LBL	Select the LBL function.
S E T = E N T / C A L L = L B L	
LBL	Select CALL to call the label.
C A L L L 0	
5 ENT	Enter the label number. Confirm your entry.
C A L L L 5 R	
4 ENT	Enter the desired number of repetitions R, for example 4. Confirm your entry. The resulting program block is CALL L 5 R4 / 4

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

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

Program blocks

0	BEGIN PGM 70	Start of program, program number
1	F 9999	High feed rate for pre-positioning
2	Z+20.000	Clearance height
5	M 3	Spindle ON, clockwise
6	X+30.000 R0	X coordinate infeed point slot
7	Y+10.000 R0	X coordinate infeed point slot
8	**LBL 1**	Start of program section 1
9	**F 150**	Machining feed rate during the program section repeat
10	**Z-12.000**	Infeed
11	**IX+16.000 R0**	Mill the slot
12	**F 9999**	High feed rate for retracting and pre-positioning
13	**Z+2.000**	Retract
14	**IX-16.000 R0**	Positioning in X
15	**IY+15.000 R0**	Positioning in Y
16	**CALL L1 R4 / 4**	Repeat program section 1 four times
17	Z+20.000	Clearance height
18	M 2	Stop program run, spindle STOP, coolant OFF
19	END PGM 70	End of program, program number

NOTES

6
Transferring Programs over the Data Interface

The TNC 122 features an RS-232-C/V.24 interface for external data storage on a device such as the HEIDENHAIN FE 401 floppy disk unit or a PC.

Programs 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 in the Technical Manual for the TNC 122.

Transferring a program to the TNC

Mode of operation: PROGRAMMING AND EDITING

PGM	Select program management.
Repeatedly ↓	To transfer from an FE 401 or PC, select PGM INPUT **FE**.. To transfer from an ME, select PGM INPUT **EXT**.
P G M I N P U T F E	
ENT	Select FE for transferring to the TNC from an FE 401, for example.
P G M N O . =	
1 2	Enter the program number, for example number 12.
ENT	Transfer the program to the TNC.

Transferring programs out of the TNC

Example: Transferring a program from the TNC to an FE 401

The TNC automatically transfers the program that you last selected for programming.

Mode of operation: PROGRAMMING AND EDITING

PGM	Select program management.
Repeatedly ↓	To transfer to an FE 401 or PC, select PGM OUTPUT **FE**. To transfer to an ME, select PGM OUTPUT **EXT**.
P G M O U T P U T F E	
ENT	Select FE for transferring from the TNC to an FE 401, for example. The TNC immediately transfers the program to the external device.

CAUTION

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

7
Executing Programs

There are two ways to run programs on the TNC:

PROGRAM RUN SINGLE BLOCK
Use the NC start key to separately start each block. It is recommended that you use SINGLE BLOCK when running a program for the first time.

PROGRAM RUN 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

⇒ Change the tool at the clearance height.
⇒ Pre-position the tool in X and Y (when the tool axis is Z).
⇒ Move the tool to the working depth.

Preparation

⇒ Clamp the workpiece to the machine table.
⇒ Set the workpiece datum.
⇒ Select the program that you wish to run.

Changing the feed rate F during program run

Some machines are equipped with a potentiometer to allow you to vary the feed rate.

Skipping program blocks

If you wish to start a program at a certain block:

⇒ Enter the block number.
⇒ Start the program as described in this chapter.

Overview of functions

Function	key
Stop machine axis movements; Interrupt program run	0
Enter the tool data	TOOL DEF

Single block

Mode of operation: PROGRAM RUN SINGLE BLOCK

For each block: I	Position for each individual program block.

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

Automatic

Mode of operation: PROGRAM RUN AUTOMATIC

I	Position. The program run indicator glows during program run.

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

Interrupting program run

To interrupt the program run, **without aborting**:

⇒ Press the NC-Stop button.
The program run indicator blinks.

To resume program run after the interruption:

⇒ Press the NC-Start button
The program run indicator glows.

To **abort** the program run

⇒ Press the NC-Stop button.
The program run indicator blinks.

⇒ Press the STOP key.
The program run indicator goes out.

To restart program run after STOP

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

To restart the program run:

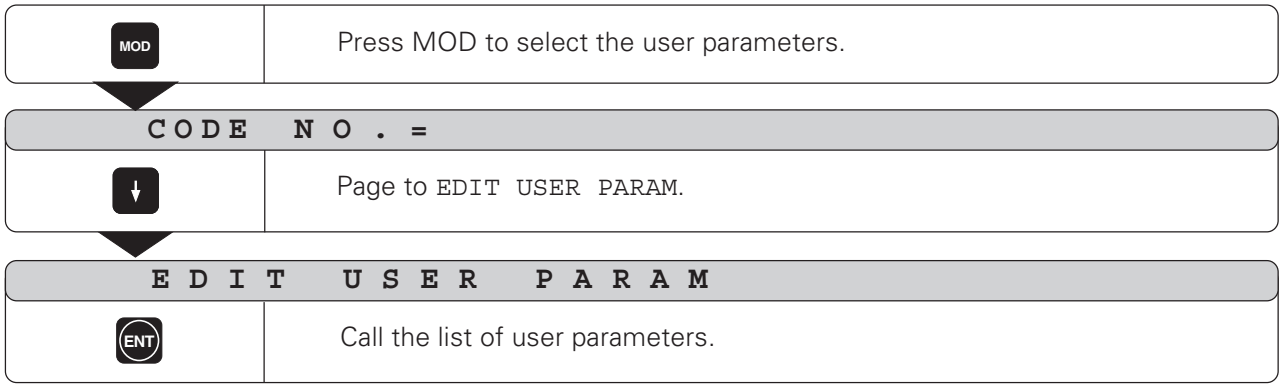
⇒ Press the NC-Start button

8
User Parameters

With **user parameters** you define the way the TNC operates in various situations. You can change user parameters without first having to enter a code number.

Selecting user parameters

Mode of operation: any



To change user parameters

⇒ Use the vertical arrow keys to select the desired user parameter.
⇒ Enter the new parameter **value**.
⇒ Confirm your entry with ENT.

To leave the user parameters

⇒ Press the DEL ☐ key to leave the user parameters.
The changes are effective **immediately**.

User Parameters in the TNC 122

Parameters whose functions are determined by the machine tool builder

The machine tool builder determines the function of the machine parameters:

- MP4310.0
- MP4310.1

Sequence for crossing the reference marks	
MP1340.0: 1st axis	X axis: ***1***
MP1340.1: 2nd axis	Y axis: ***2***
MP1340.2: 3rd axis	Z axis: ***3***
	No reference mark evaluation: ***0***

Programming station setup	
MP7210	TNC with machine: ***0***
	TNC as a programming station with active PLC: ***1***
	TNC as programming station with inactive PLC: ***2***

Dialog language	
MP7230	German: ***0***
	English: ***1***
	French: ***2***
	Italian: ***3***
	Spanish: ***4***

Position display in the tool axis	
MP7285	Display the position of the tool datum: ***0***
	Display the position of the tool point: ***1***

Select the position display	
MP7322	Actual position: ***0***
	Servo lag: ***1***
	Reference position: ***2***

Enable continuing traverse with the direction keys	
MP7680	Continuing traverse disabled: ***0***
	Continuing traverse enabled: ***1***

9
Tables and Overviews

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

- Miscellaneous functions (M functions) with predetermined effect
- Vacant miscellaneous functions
- Frequently occurring display messages and their meanings
- Technical information
- Accessories: FE 401 floppy disk unit

Miscellaneous functions (M functions)

M functions with predetermined effect

With the M functions the TNC controls:

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

The machine tool builder determines which miscellaneous functions are available on your TNC and what effect they have.

M number	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

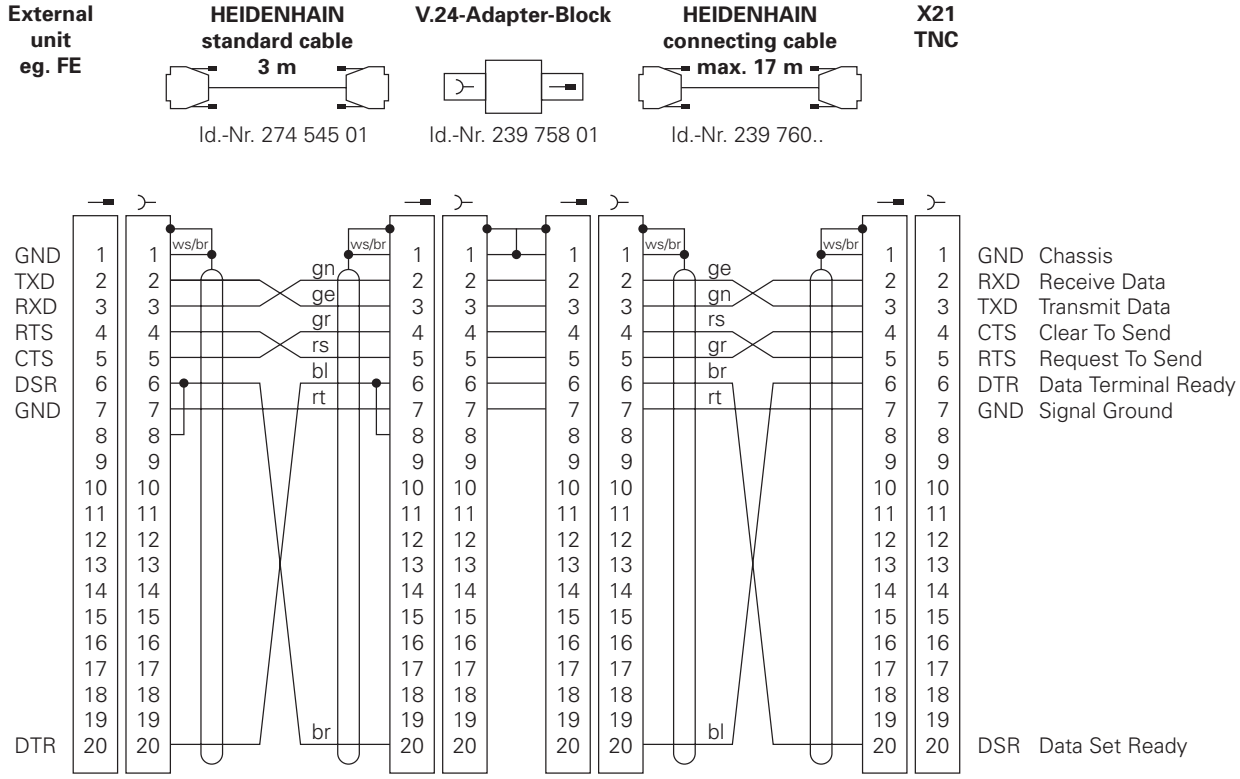
The machine manufacturer can inform you of the tasks he has assigned to the vacant miscellaneous functions listed on this page.

M number	Vacant miscellaneous 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 miscellaneous 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

HEIDENHAIN devices



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

The X21 interface complies with the recommendations in VDE 0160, 5.88 for separation from line power.

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.

TNC Messages

The TNC generates error messages automatically. They appear, among other things, whenever the TNC detects

- incorrect data input and
- logical errors in the program

To **clear** a message from the screen:

⇒ Press the CL key.

The following are the most frequent messages and their remedies.

0 NOT ALLOWED

Do not program CALL L.

ENTRY INCORRECT

- Enter a correct LBL number.
- Enter a value within permissible limits

EXT. NOT READY

Correctly connect the external device.

LBL ALREADY USED

A given label number can be assigned only once.

LBL NOT FOUND

Set the label before calling it.

NESTING TOO DEEP

Program sections and subprograms cannot be nested more than 8 times.

NOT CURRNT BLOCK

Start the program run at the BEGIN block.

PGM MEM OVERFLOW

You have overloaded the TNC's storage capacity.

WRONG AXIS PROGRAMMED

Do not program axes that are not controlled by the TNC.

Specifications

TNC technical data	
Brief description	Compact straight cut control, mechanically and electrically compatible with the TNC 121 straight cut control; with analog speed control for machines with up to 3 axes with central drive
Program memory	20 part programs 500 program blocks 500 program blocks per program
Position data	Single-axis Cartesian coordinates absolute or incremental
Unit of measurement	Millimeters
Display step	Depending on encoders and machine parameters, for example 0.005 mm for a grating period of 20 µm
Input range	0.005 mm to 9 999.999 mm
Max. range of traverse	+/– 10 000 mm
Maximum feed rate	Machining: 10 000 mm/min Rapid traverse: 30 000 mm/min
Data interface	RS-232-C/V.24
Data transfer rate	FE setting: 9600 baud EXT setting: 2400 baud
Program routines	Subprograms, Program section repeats
Fixed cycles	Bolt hole circles; Linear hole patterns
Ambient temperature	Operation: 0° C to 45° C (32° to 113° F) Storage: –30° C to 70° C (–22° to 158° F)
Weight	Approx. 3 kg
Power consumption	Approx. 19 W

Accessory

FE 401 Floppy Disk Unit	
Description	Portable bench-top unit
Data interface	Two RS-232-C/V.24
Data transfer rate	TNC setting: 2400 baud to 38 400 baud PRT setting: 110 baud to 9600 baud
Disk drives	Two drives, one for copying
Floppy disks	3,5'', DS, DD, 135 TPI
Memory capacity	795 kilobytes (approx. 25 000 program blocks), 256 files

Sequence of Program Steps
Milling an outside contour

Mode of Operation: PROGRAMMING AND EDITING

Program step		
1	**Select program**	
	Entries:	Program number Unit of measurement in program
2	**Call tool data**	
	Entries:	Tool number Spindle axis
3	**Tool change**	
	Entries:	Coordinates of 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:	Coordinate of the (first) working depth Feed rate (rapid traverse)
6	**Approach 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	**Approach 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**	

291 903 20 · SW01 · 3 · 12/94 · F+W · Printed in Germany · Subject to change without notice