**HEIDENHAIN**

User's Manual
ISO Programming

TNC 426
TNC 425
TNC 415 B
TNC 407

Controls on the TNC 426, TNC 425, TNC 415 B and TNC 407

**Controls on the visual display unit**

|	
⟳ ▭ ▭	Toggle display between machining and programming modes
⟳ GRAPHICS TEXT SPLIT SCREEN	Split screen layout
▢	Soft keys for selecting functions in screen
◁ ▷	Shift keys for the soft keys
☀ ◐	Brightness, contrast

Typewriter keyboard for entering letters and symbols

Q W E R T Y	File names/ comments
G F S T M ...	ISO programming

Machine operating modes

✋	MANUAL OPERATION
⊕	EL. HANDWHEEL
▣	POSITIONING WITH MDI
▣	PROGRAM RUN/SINGLE BLOCK
➜	PROGRAM RUN/FULL SEQUENCE

Programming modes

◈	PROGRAMMING AND EDITING
➔	TEST RUN

Program and file management

PGM MGT PGM NAME	Select programs and files
CL PGM	Delete programs and files *(not on TNC 426)*
PGM CALL	Enter program call in a program
EXT	External data transfer *(not on TNC 426)*
MOD	MOD functions
CALC	Pocket calculator *(TNC 426 only)*

Moving the cursor and going directly to blocks, cycles and parameter functions

← ↑ → ↓	Move the cursor (highlight)
GOTO	Go directly to blocks, cycles and parameter functions

Override control knobs

Feed rate   Spindle speed

100   100
50 — 150   50 — 150
0 ⋁⋁⋁ F %   0 ↻ S %
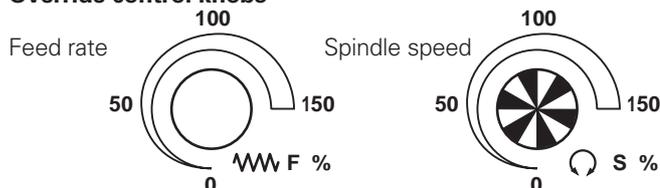
Programming path movements
(conversational programming only)

APPR DEP	Approach/depart contour
L	Straight line
CC	Circle center/pole for polar coordinates
C	Circle with center
CR	Circle with radius
CT	Tangential circle
CHF	Chamfer
RND	Corner rounding

Tool functions *(conversational programming only)*

TOOL DEF TOOL CALL	Enter or call tool length and radius
R+ᴿ R-ᴸ	Activate tool radius compensation *(not on TNC 426)*

Cycles, subprograms and program section repeats
(conversational programming only)

CYCL DEF CYCL CALL	Define and call cycles
LBL SET LBL CALL	Enter and call labels for subprogramming and program section repeats
STOP	Enter program stop in a program
TOUCH PROBE	Enter touch probe functions in a program

Coordinate axes and numbers, editing

X .. V	Select coordinate axes or enter them into a program
0 ... 9	Numbers
•	Decimal point
⁻/+	Arithmetic sign
P	Polar coordinates
I	Incremental dimensions
Q	Q parameters for part families or mathematical functions
✛	Capture actual position
NO ENT	Skip dialog questions, delete words
ENT	Confirm entry and resume dialog
END	End block
CE	Clear numerical entry or TNC message
DEL	Abort dialog, delete program sections

TNC Guideline
From the workpiece drawing to program-controlled machining

Step	Task	TNC operating mode	Section in manual
	Preparation		
1	Select tools	——	——
2	Set workpiece datum for coordinate system	——	——
3	Determine spindle speeds and feed rates	——	——
4	Switch on the machine	——	1.3
5	Cross over reference marks	🖐 or ⊕	1.3, 2.1
6	Clamp workpiece	——	——
7	Set datum / Reset position display ...		
7a	... **with** 3D touch probe	🖐 or ⊕	2.5
7b	... **without** 3D touch probe	🖐 or ⊕	2.3
	Entering and testing part programs		
8	Enter part program or download over external data interface	◈ or EXT	5 to 8, 9
9	Test part program for errors	➡	3.1
10	Test run: Run the program block by block without tool	🅑➡	3.2
11	Optimize the part program (if necessary)	◈	5 to 8
	Machining the workpiece		
12	Insert tool and run program	➡	3.2

How to use this manual

This manual describes functions and features available on TNCs as of the following NC software numbers:

	NC Software No.
TNC 407	**280 580 04**
TNC 415 B, TNC 425	**280 540 04**
TNC 415 F, TNC 425 E	**280 560 04**
TNC 426 CA, TNC 426 PA	**280 462 01**
TNC 426 CE, TNC 426 PE	**280 482 01**

The suffixes E and F indicate export versions of the TNC.

The export versions TNC 415 F, TNC 425 E, TNC 426 CE, and
TNC 426 PE have the following limitations:

• Input and machining accuracy are limited to 1 µm
• Simultaneous linear movement in up to 4 axes

Some of the functions described in this manual are not available on all
TNCs. These functions are marked with symbols:

407 Function not available on the TNC 407

415 Function not available on the TNC 415

425 Function not available on the TNC 425

426 Function not available on the TNC 426

The machine manufacturer adapts the features offered by the TNC to the
capabilities of the specific machine tool by setting machine parameters.
This means that not every machine tool will have all of the functions
described in this manual.

Some of the TNC functions which are not available on every machine are:

• Probe functions for the 3D touch probe
• Digitizing option *(conversational programming only)*
• Measuring tools with the TT 120 touch probe
  (conversational programming only)
• Rigid tapping
• Re-approaching a contour after an interruption

Your machine manual provides more detailed information. If you think a
function may be unavailable because of a defect, please contact the
machine tool builder.

Many machine manufacturers and HEIDENHAIN offer programming
courses for the TNCs. We recommend these courses as an effective way
of improving your programming skill and sharing information and ideas with
other TNC users.

This manual is intended both for the TNC beginner and the TNC expert.

The **TNC beginner** can use it as a step-by-step workbook. The manual begins with an explanation of the basics of numerical control (NC) and provides a glimpse into their application in the TNC. It then introduces the technique of conversational programming. All of the examples can be practiced directly on the TNC. Each function is explained thoroughly when it is used for the first time.

The TNC beginner should work through this manual completely from beginning to end to ensure that he is capable of fully exploiting the features of this powerful tool.

The **TNC expert** can use the manual as a comprehensive review and reference work. The table of contents and numerous cross references help him quickly find the topics and information he needs. Easy-to-read dialog flowcharts show him how to enter data for the desired function.

The dialog flowcharts aid the beginner by providing a description of the function of each key in a box to its right. If the user already knows the keys, he can concentrate on the illustrated input overview at the left of the flowchart. The TNC dialog messages are represented in shaded boxes above the answering input sequence.

Layout of the dialog flowcharts

Dialog initiation key

```
G  8  3  ▶
```

DIALOG PROMPT (ON THE TNC SCREEN)

e.g. **8** **3** **ENT**

Answer the prompt with these keys

Here the manual explains the function of the keys.

NEXT DIALOG PROMPT

┼

Press this key

Function of the key

‾/+

Or this key

Function of the alternative key

A broken line indicates that either the key above it or below it can be pressed.
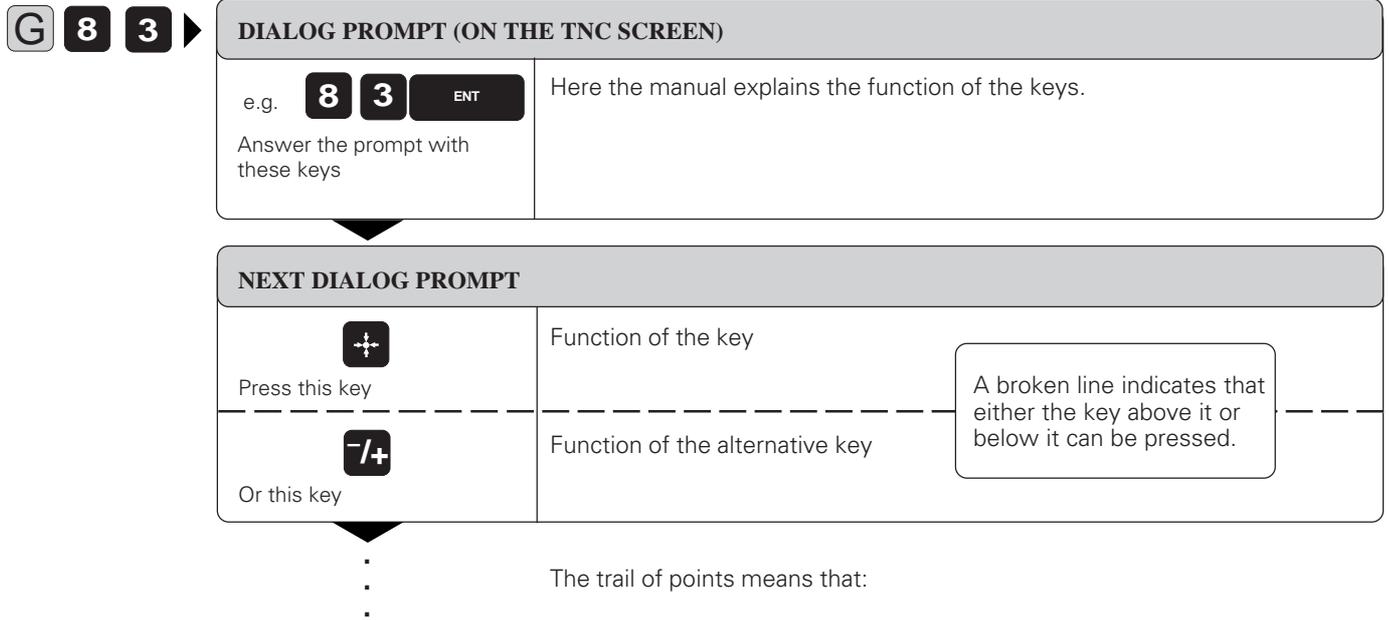
The trail of points means that:

• the dialog is not completely illustrated, or
• the dialog continues on the next page.

Abbreviated dialog flowcharts

In abbreviated flowcharts an arrow (➤) is used to indicate new entries or work steps.

Contents User's Manual TNC 407, TNC 415 B, TNC 425, TNC 426 (280 5x0-xx, 280 462-xx) ISO Programming

1 Introduction

2 Manual Operation and Setup

3 Test Run and Program Run

4 Programming

5   Programming Tool Movements

6 Subprograms and Program Section Repeats

7 Programming with Q Parameters

8 Cycles

9 External Data Transfer

10 MOD Functions

11 Tables, Overviews and Diagrams

1.1 The TNC 400 Series

The TNCs are shop-floor programmable contouring controls for boring machines, milling machines and machining centers with up to 5 axes. They also feature oriented spindle stop.

Two operating modes are always active simultaneously: one for machine movements (machining modes) and one for programming or program testing (programming modes).

TNC 426

The TNC 426 PA features digital control of machine axis speed. This provides high geometrical accuracy, even with complex workpiece surfaces and at high machining speeds.

An integrated 170 megabyte hard disk provides storage for programs that were created on external devices. The TNC 426 also offers an on-screen pocket calculator.

TNC 425

The TNC 425 also features digital control of machine axis speed. This results in high geometrical accuracy, even with complex workpiece surfaces and at high machining speeds.

TNC 415 B

The TNC 415 B uses an analog method of speed control in the drive amplifier. All the programming and machining functions of the TNC 425 are also available on the TNC 415 B.

TNC 407

The TNC 407 uses an analog method of speed control in the drive amplifier. Some functions are not available on the TNC 407, such as:

• Graphics during program run
• Tilting the machining plane
• Linear movement in more than three axes

Technical differences between the TNCs

	TNC 426 PA	TNC 426 CA	TNC 425	TNC 415 B	TNC 407
Speed control	Digital	Analog	Digital/analog	Analog	Analog
Block processing time	4 ms	4 ms	4 ms	4 ms	24 ms
Control loop cycle time: Contouring interpolation	3 ms	3 ms	3 ms	2 ms	6 ms
Control loop cycle time: Fine interpolation	0.6 ms	---	0.6 ms	0.6 ms	---
Program memory	170 M byte (hard disk)	170 M byte (hard disk)	256 K byte	256 K byte	128 K byte
Input resolution	0.1 µm	0.1 µm	0.1 µm	0.1 µm	1 µm

Visual display unit and keyboard

The 14-inch color monitor displays all the information necessary for effective use of the TNC's capabilities.

The keys are grouped on the keyboard according to function. This makes it easier to create programs and to use the TNC's functions.

Programming

The TNCs are programmed in ISO format.

It is also possible to program in easy-to-understand HEIDENHAIN conversational format (a separate User's Manual is available for this).

Graphics

Workpiece machining can be graphically simulated both during machining (except on TNC 407) or before actual machining. Various display modes are available.

Compatibility

The TNCs can execute all part programs written on HEIDENHAIN TNC 150 B controls or later.

Keyboard

The keys on the TNC keyboard are marked with symbols and abbreviations that make them easy to remember. They are grouped according to the their functions. The functions of the individual keys are described in the front cover fold-out of the TNC user's manual. A description of machine panel buttons is provided in the manual for your machine tool.

The keyboard of TNC 407, TNC 415 and TNC 425 controls

Typewriter-style keyboard for entering file names, comments and other texts, as well as programming in ISO format

Numerical input and axis selection

Program and file management

Arrow keys and GOTO key

Machine operating modes

Programming modes

Dialog initiation for conversational programming
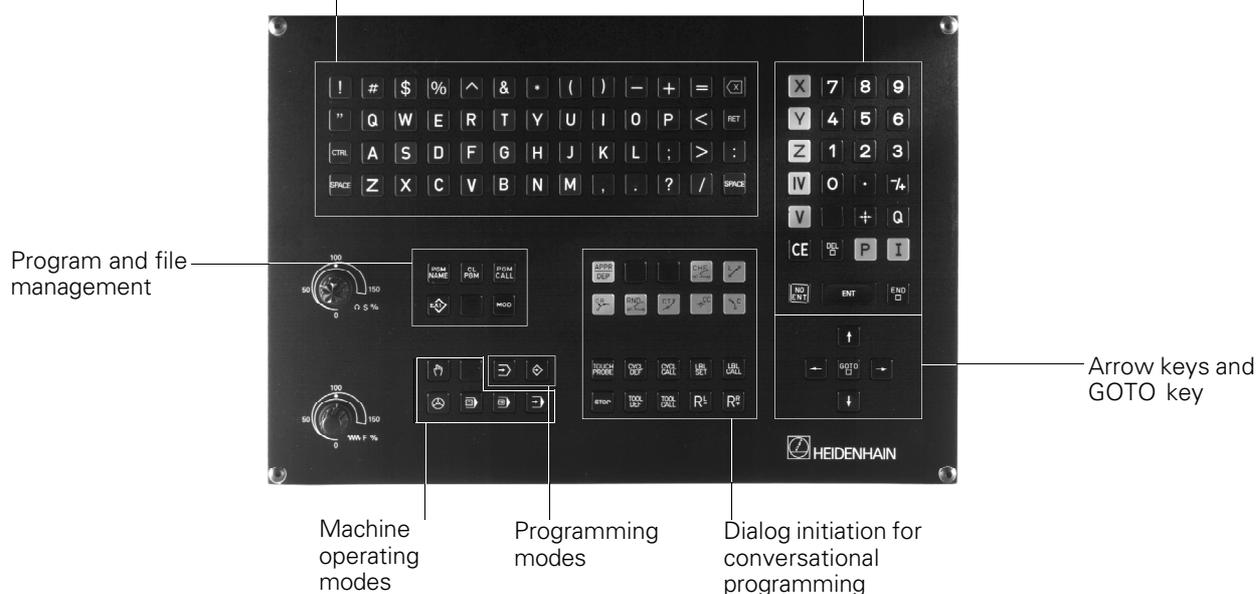
The keyboard of TNC 426 controls

Typewriter-style keyboard for entering file names, comments and other texts, as well as programming in ISO format

Numerical input and axis selection

File management, pocket calculator, MOD functions, HELP functions

Arrow keys and GOTO key

Machine operating modes

Programming modes

Dialog initiation

Visual display unit

Brightness control

Contrast control

Switchover between
the active program-
ming and machining
modes

GRAPHICS
TEXT
SPLIT
SCREEN

SPLIT SCREEN key
for switching screen
layout (see page 1-6)

Soft keys with context-specific
functions, and two shift keys
for additional soft-key rows
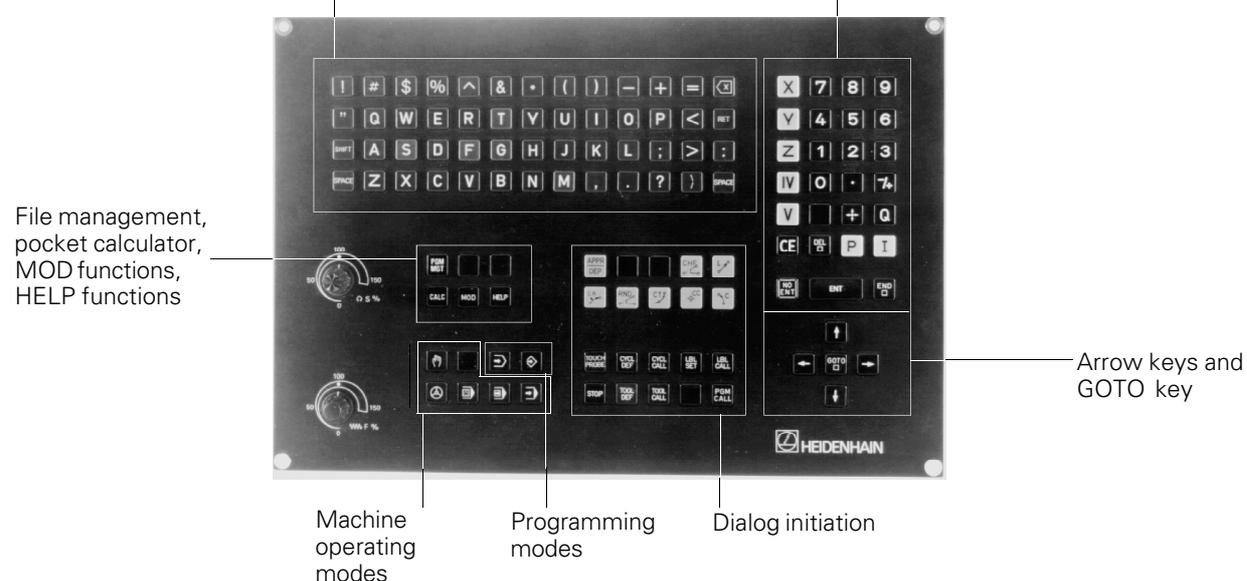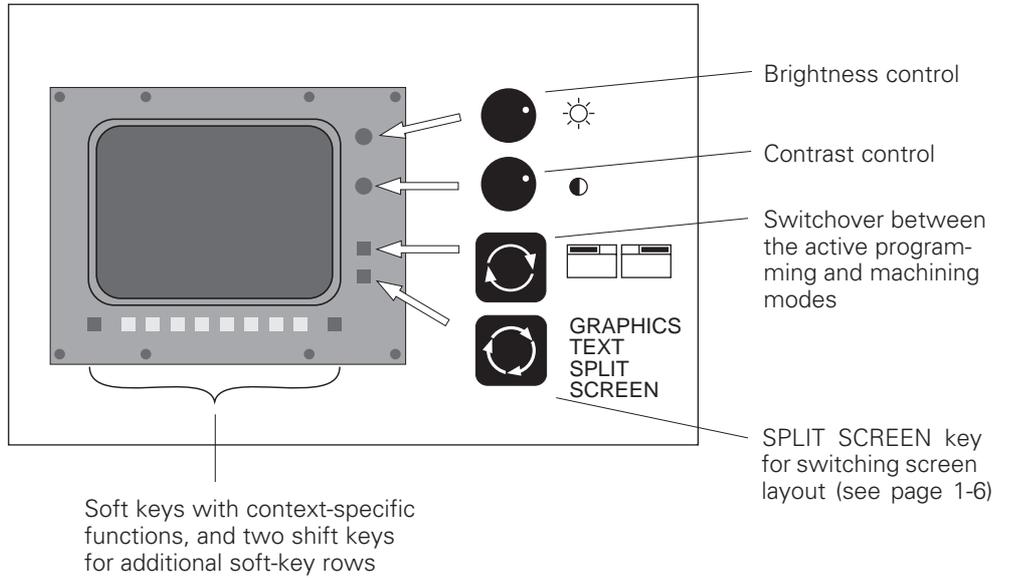
Headline

The two selected TNC modes are shown in the screen headline:
the machining mode to the left and the programming mode to the right.
The currently active mode is displayed in the larger box, where dialog
prompts and TNC messages also appear.

Soft keys

The soft keys select the functions shown in the soft-key row immediately
above them. The shift keys to the right and left call up additional soft-key
rows. Colored lines above the soft-key row indicate the number of
available rows. The line representing the active row is highlighted.

1.1   The TNC 400 Series

Screen layout

You can select the type of display on the TNC screen by pressing the SPLIT SCREEN key and one of the soft keys listed below. Depending on the active mode of operation, you can select:

Mode of operation	Screen layout	Soft key
MANUAL ELECTRONIC HANDWHEEL	Positions	POSITION
	Left: positions Right: STATUS	POSITION + STATUS
POSITIONING WITH MDI	Program blocks	PGM
	Left: program blocks Right: STATUS	PGM + STATUS
PROGRAM RUN/FULL SEQUENCE PROGRAM RUN/SINGLE BLOCK TEST RUN	Program blocks	PGM
	Left: program blocks Right: program structure (*conversational programming only*)	PGM + SECTION
	Left: program blocks Right: STATUS	PGM + STATUS
	Left: program blocks Right: graphics	PGM + GRAPHICS
	Graphics	GRAPHICS
PROGRAMMING AND EDITING	No screen selection possible, the TNC displays program blocks only	

Screen layout of modes

PROGRAMMING AND EDITING:

Machining
mode

Programming mode is selected

Text of the
selected
program

Soft-key row

```
MANUAL     PROGRAMMING AND EDITING
OPERATION
N20 G01 Z-25 F250 *
N20 G83 P01 -1 P02 -25 P03 -5 P04 0
    P05 150 *
N20 X+10 Y+52 M3 *
N20 M99 *
N20 X+25 Y+90 M99 *
N20 G98 L1 *
N20 G91 X+10 M99 *
N20 L1,7 *
N20 G90 X+100 Y+50 *
N20 G00 Z+100 *
N20 X+150 Y+25,5 *
N20 G84 P01 -2 P02 -20 P03 0 P04 250 *
N20 Z+2 M3 *
N20 M99 *

PARA-
METER
```

TEST RUN:

Machining
mode

Programming mode is selected

Text of the
selected
program

Graphics
(or additional
status display)

Soft-key row

```
PROGRAM RUN   TEST RUN
FULL SEQUENCE

%3813 G71 *
N10 D00 Q1 P01 +0 *
N20 D00 Q2 P01 +0 *
N30 D00 Q3 P01 +0 *
N35 D00 Q6 P01 +40 *
N36 D00 Q16 P01 +10 *
N40 D00 Q7 P01 +90 *
N50 D00 Q17 P01 +270 *
N60 D00 Q8 P01 +0 *
N70 D00 Q18 P01 +90 *
N80 D00 Q9 P01 +0 *
N90 D00 Q10 P01 +50 *
N100 D00 Q12 P01 +0 *
N110 *
N120 D00 Q20 P01 +500 *

                          X  +5,5  V  +6,5
                                     00:16:45

              /□    START   STOP          RESET
         OFF/ON  SINGLE   AT    START    +
              □     N           START
```

MANUAL OPERATION and ELECTRONIC HANDWHEEL modes:

A machining mode is selected

Programming mode

```
MANUAL OPERATION                          PROGRAMMING
                                          AND EDITING


ACTL.
      X  +1235,0000        NOML.
      Y   +456,0000        X +1235,0000
      Z    +48,0000        Y  +456,0000
      B    +12,0000        Z   +48,0000
      C   +102,0000        B   +12,0000
                           C  +102,0000


                                          B +15,0000
                                          C +30,0000
T
M 5/9      F 0                   BASIC ROTATION


  M     S    TOUCH   DATUM         3D ROT        TOOL
             PROBE    SET                        TABLE
```

- Coordinates
- Selected axis
- ✳ means TNC in operation
- Status display, e.g. feed rate F, miscellaneous function M, symbols for basic rotation and/or tilted working plane

Additional status display

Soft-key row

PROGRAM RUN/FULL SEQUENCE, PROGRAM RUN/SINGLE BLOCK

A machining mode is selected

Programming mode

```
PROGRAM RUN / SINGLE BLOCK                PROGRAMMING
                                          AND EDITING

%3805 G71 *
N10 G30 G17 X+0 Y+0 Z-40 *
N20 G31 G90 X+100 Y+100 Z+0 *
N30 G99 T1 L+0 R+7,5 *
N40 T1 G17 S1500 *
N50 G00 G40 G90 Z+50 *
N60 G75 P01 -2 P02 -20 P03 -10 P04 100
    P05 X+80 P06 Y+50 P07 500 *
N70 X+50 Y+50 M3 *

                          27°              00:00:00
ACTL.      X  +132,6870     Y    +12,5600
           Z  +160,2560     B    +30,0000
           C   +90,0000
T                          F 0        M 5/9

                     RESTORE        /□
                     POS. AT        OFF/ON
                        N
```

Text of the selected program

Graphics (or additional status display, or program structure)

Status display

Soft-key row

TNC Accessories

3D Touch Probe Systems

The TNC provides the following features when used in conjunction with a HEIDENHAIN 3D touch probe:

- Electronic workpiece alignment (compensation of workpiece misalignment)
- Datum setting
- Measurement of the workpiece during program run
- Digitizing 3D surfaces (optional, only available with conversational programming)
- Measuring tools with the TT 120 touch probe (only available)

Fig. 1.6:   TS 220 and TS 630 3D-touch probes

Electronic Handwheels

Electronic handwheels facilitate precise manual control of the axis slides. Similar to 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.

Portable handwheels such as the HR 330 are connected via cable to the TNC. Integral handwheels such as the HR 130 are built into the machine control panel. An adapter permits connection of up to three handwheels.

Your machine manufacturer can tell you more about the handwheel configuration of your machine.

Fig. 1.7:   HR 330 electronic handwheel

1.2 Fundamentals of NC

Introduction

This chapter discusses the following topics:

- What is NC?
- The part program
- Programming
- Reference system
- Cartesian coordinate system
- Additional axes
- Polar coordinates
- Setting the pole
- Datum setting
- Absolute workpiece positions
- Incremental workpiece positions
- Programming tool movements
- Position encoders
- Reference marks

What is NC?

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

The part program

The part program is a complete list of instructions for machining a part. It contains such information as the target position of a tool movement, the path function (how the tool should move toward the target position) and the feed rate. Information on the radius and length of the tool, spindle speed and tool axis must also be included in the program.

Programming

ISO programming is partially dialog-guided. The programmer is free to enter the individual commands (words) in each block in any sequence (except with G90/G91). The commands are automatically sorted by the TNC when the block is concluded.

1.2   Fundamentals of NC

Reference system

In order to define positions, a reference system is necessary. For example, positions on the earth's surface can be defined "absolutely" by their geographic coordinates of longitude and latitude. The word *coordinate* comes from the Latin word for "that which is arranged." The network of horizontal and vertical lines around the globe constitute an absolute reference system — in contrast to the relative definition of a position that is referenced to a known location.
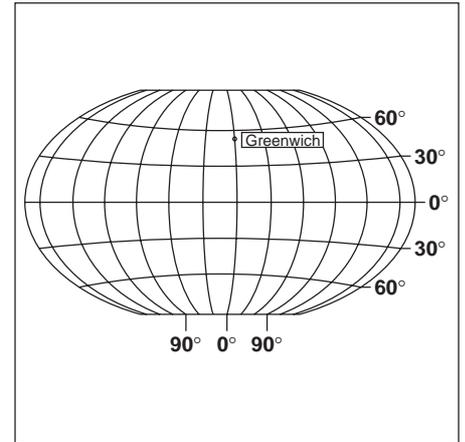
Fig. 1.8:   The geographic coordinate system is an absolute reference system

Cartesian coordinate system

On a TNC-controlled milling machine, workpieces are normally machined according to a workpiece-based Cartesian coordinate system (a rectangular coordinate system named after the French mathematician and philosopher Renatus Cartesius, who lived from 1596 to 1650). The Cartesian coordinate system is based on three coordinate axes 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.
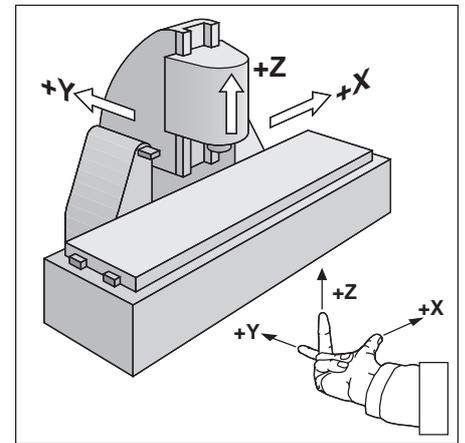
Fig. 1.9:   Designations and directions of the axes on a milling machine

Additional axes

The TNC can control the machine in more than three axes. Axes **U**, **V** and **W** are secondary linear axes parallel to the main axes X, Y and Z, respectively (see illustration). **Rotary axes** are also possible, and are designated as **A**, **B** and **C**.
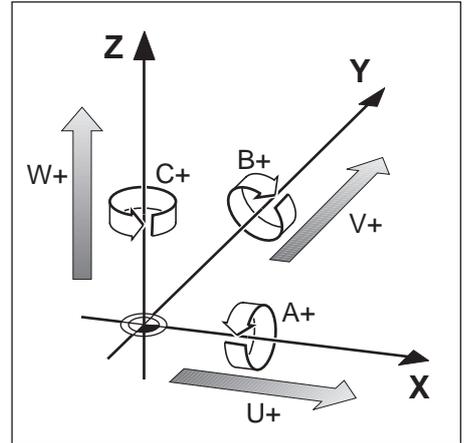
Fig. 1.10:   Direction and designation of additional axes

Polar coordinates

Although the Cartesian coordinate system is especially useful for parts whose dimensions are mutually perpendicular, in the case of parts containing circular arcs or angles it is often simpler to give the dimensions in polar coordinates. While Cartesian coordinates are three-dimensional and can describe points in space, polar coordinates are two-dimensional and describe points in a plane.

Polar coordinates have their datum at a **pole I, J, K** from which a position is measured in terms of its distance from the pole and the angle of its position in relation to the pole.

Fig. 1.11:   Identifying positions on a circular arc with polar coordinates

You could think of polar coordinates as the result of a measurement using a scale whose zero point is fixed at the datum and which you can rotate to different angles in the plane around the pole.

The positions in this plane are defined by the

• **Polar Radius R**, the distance from the circle center I, J to the position, and the
• **Polar Angle H**, the size of the angle between the reference axis and the scale.

Setting the pole

The pole is set by entering two Cartesian coordinates. These coordinates also determine the reference axis for the polar angle H.

Coordinates of the pole	Angle reference axis
I J	+X
J K	+Y
K I	+Z
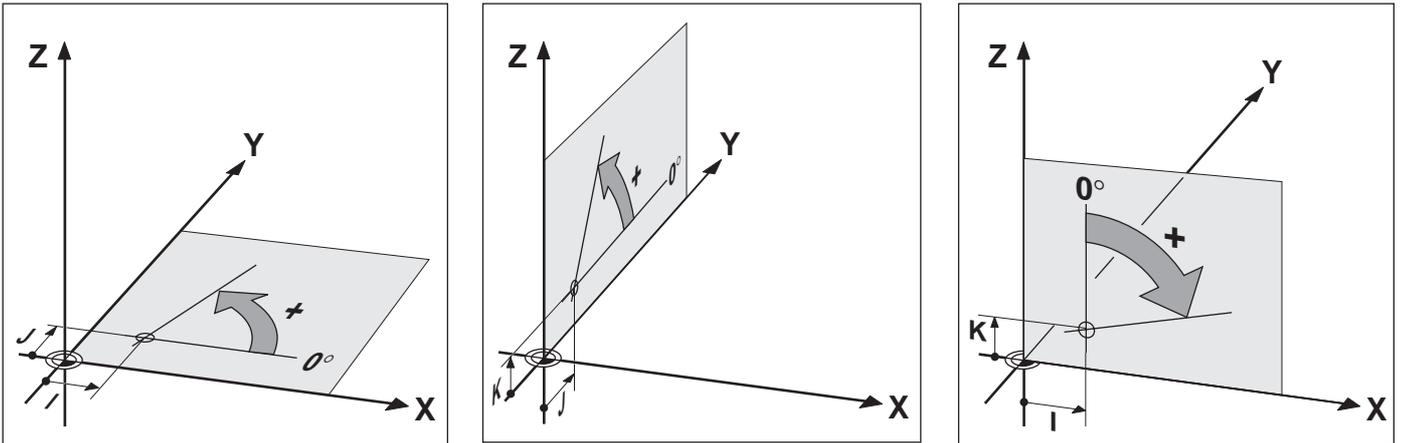
Fig. 1.12:   Polar coordinates and their associated reference axes

Datum setting

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 system. 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).
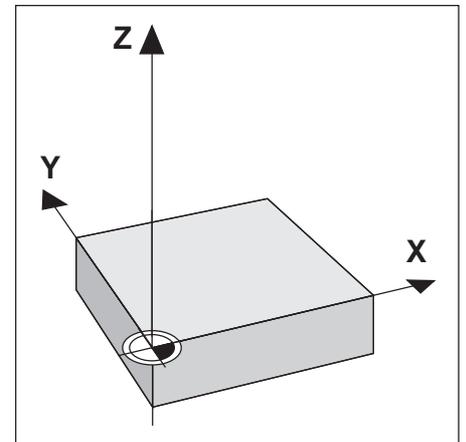
Fig. 1.13:   The workpiece datum represents the origin of the Cartesian coordinate system

Example:

Drawing with several relative datums
(ISO 129 or DIN 406 Part 11, fig. 171)
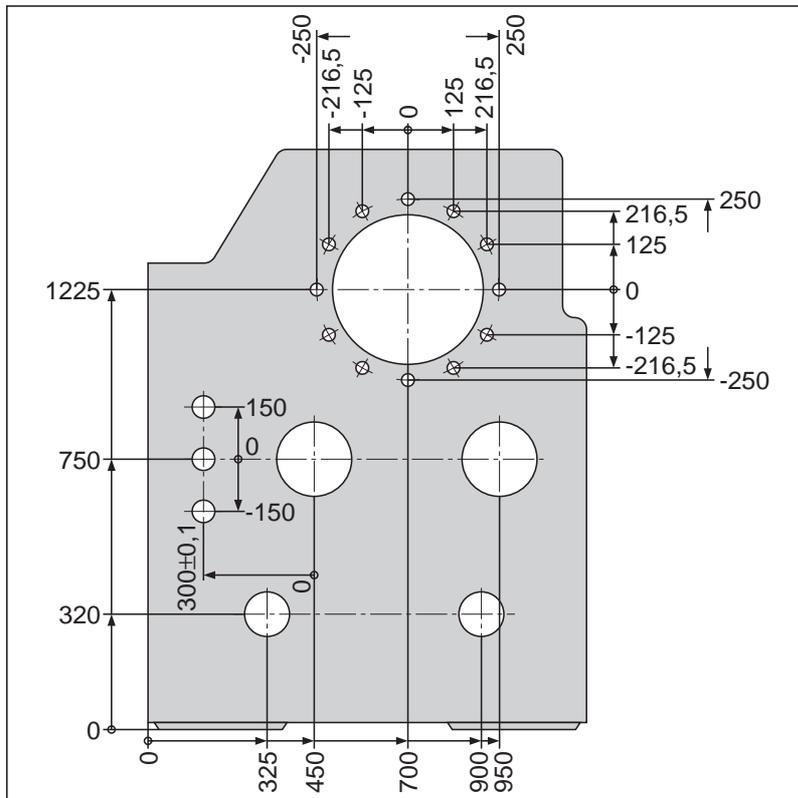
Example:

Coordinates of point ① :

X = 10 mm
Y =   5 mm
Z =   0 mm

The datum of the Cartesian coordinate system is located 10 mm from
point ① on the X axis and 5 mm from it on the Y axis.

The 3D Touch Probe System from HEIDENHAIN is an especially
convenient and efficient way to find and set datums.
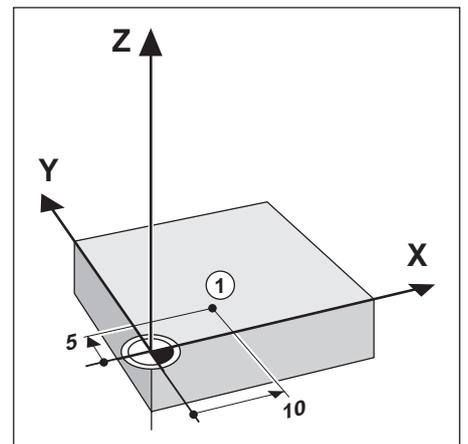
Fig. 1.14:   Point ① defines the coordinate
system

Absolute workpiece positions

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

Example:
Absolute coordinates of position ①:
X = 20 mm
Y = 10 mm
Z = 15 mm

If you are drilling or milling a workpiece according to a workpiece drawing with absolute coordinates, you are moving the tool **to** the value of the coordinates.
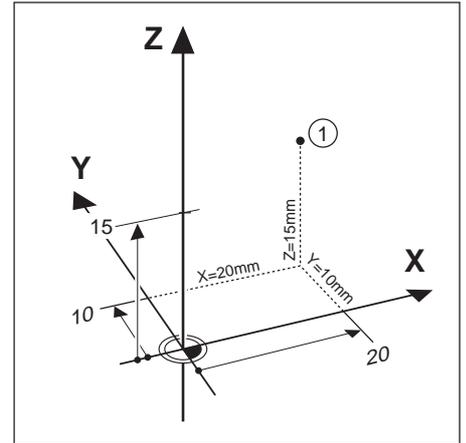
Fig. 1.15:   Position definition through absolute coordinates

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 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.

An incremental position definition is therefore a specifically *relative* definition. This is also the case when a position is defined by the distance-to-go to the nominal position. The distance-to-go has a negative sign if the target position lies in the negative axis direction from the actual position.
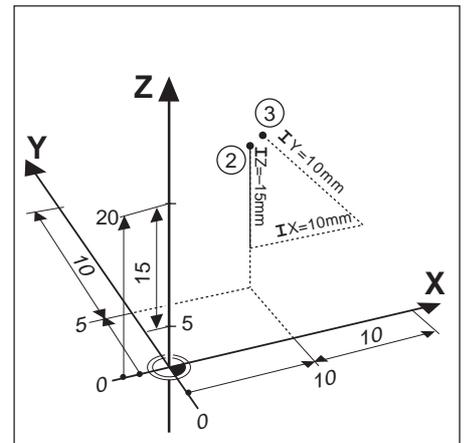
Fig. 1.16:   Position definition through incremental coordinates

1.2   Fundamentals of NC

The polar coordinate system can also express both types of dimensions:

- **Absolute polar coordinates** always refer to the pole (I, J) and the reference axis.
- **Incremental polar coordinates** always refer to the last nominal position of the tool.
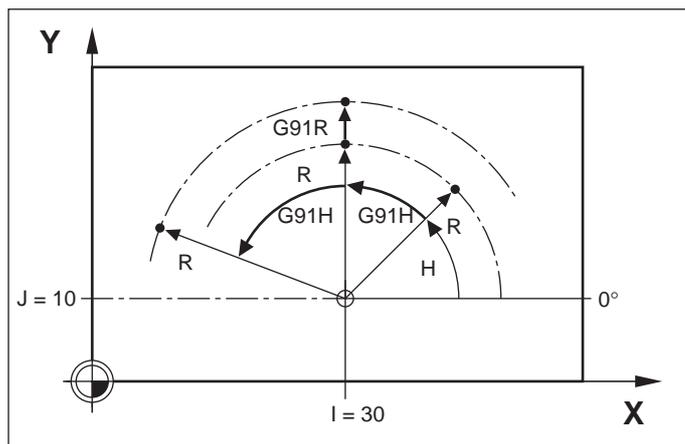
Fig. 1.17:   Incremental dimensions in polar coordinates (designated by G91)

Example:

**Workpiece drawing with coordinate dimensioning
(according to ISO 129 or DIN 406, Part 11; figure 179)**
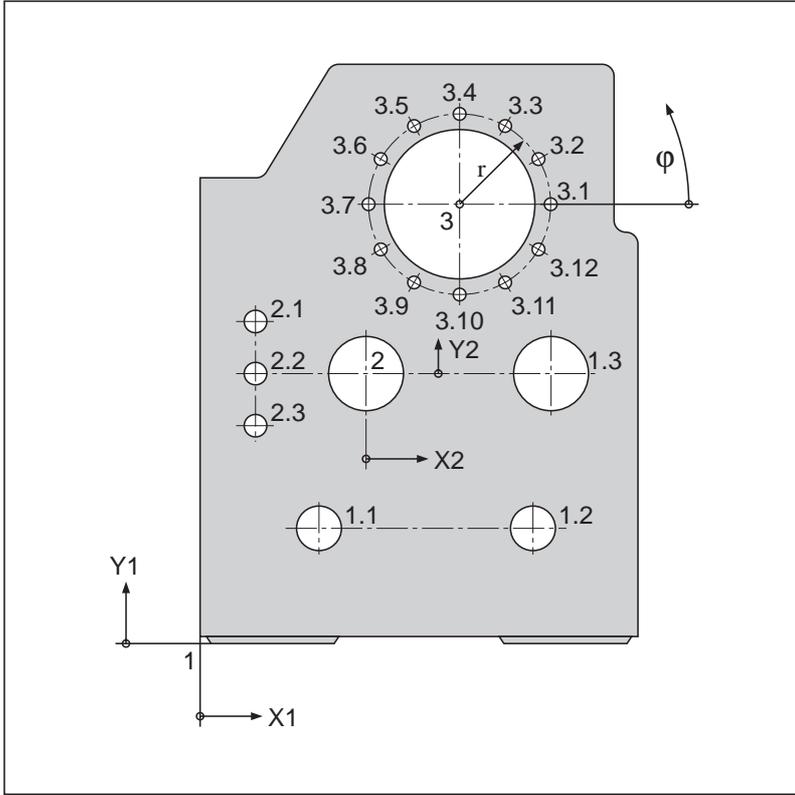
Coordinate origin	Pos.	Dimensions in mm				
		Coordinates				
		X1 X2	Y1 Y2	r	φ	d
1	1	0	0			–
1	1.1	325	320			Ø 120 H7
1	1.2	900	320			Ø 120 H7
1	1.3	950	750			Ø 200 H7
1	2	450	750			Ø 200 H7
1	3	700	1225			Ø 400 H8
2	2.1	–300	150			Ø 50 H11
2	2.2	–300	0			Ø 50 H11
2	2.3	–300	–150			Ø 50 H11
3	3.1			250	0°	Ø 26
3	3.2			250	30°	Ø 26
3	3.3			250	60°	Ø 26
3	3.4			250	90°	Ø 26
3	3.5			250	120°	Ø 26
3	3.6			250	150°	Ø 26
3	3.7			250	180°	Ø 26
3	3.8			250	210°	Ø 26
3	3.9			250	240°	Ø 26
3	3.10			250	270°	Ø 26
3	3.11			250	300°	Ø 26
3	3.12			250	330°	Ø 26

Programming tool movements

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

👆 You always program as if the tool moves and the workpiece remains stationary.

If the machine table moves, the corresponding axes are identified on the machine operating panel with a prime mark (e.g., X′, Y′). The programmed direction of such axis movement always corresponds to the direction of tool movement relative to the workpiece but in the opposite direction.
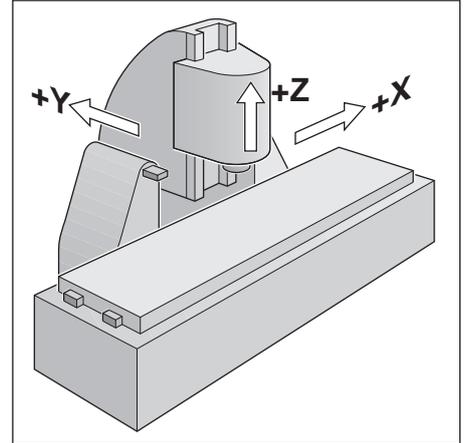
Fig. 1.18:   On this machine the tool moves in the Y and Z axes, and the table moves in the +X' axis.

Position encoders

Position encoders convert the movement of the machine axes into electrical signals. The control constantly evaluates these signals to calculate 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.
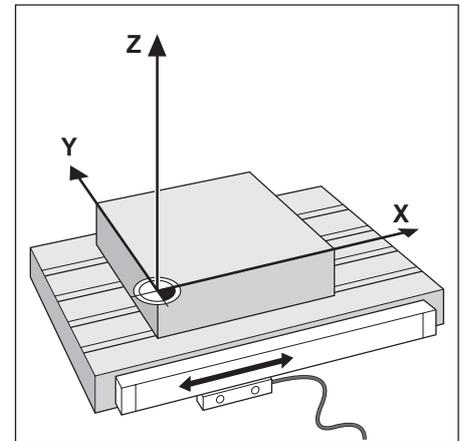
Fig. 1.19:   Linear position encoder, here for the X axis

Reference marks

The scales of the position encoders contain one or more reference marks. When a reference mark is crossed over, it generates a signal which identifies that position as the machine axis reference point. With the aid of this reference mark the TNC can re-establish the assignment of displayed positions 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.
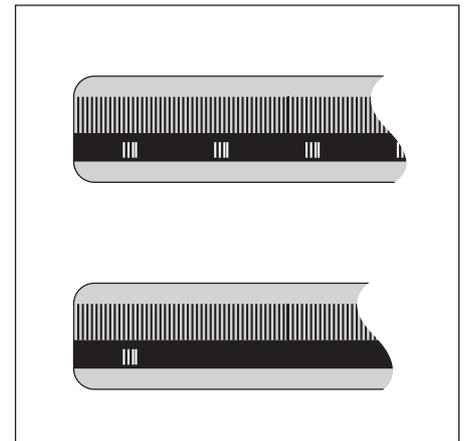
Fig. 1.20:   Linear scales: with distance-coded reference marks *(upper illustration)* and one reference mark *(lower illustration)*

1.3 Switch-On

Switch-on and traversing the reference points can vary depending on the individual machine tool. Your machine manual provides more information on these functions.

Switch on the TNC and machine tool. The TNC automatically initiates the following dialog:

MEMORY TEST
The TNC memory is automatically checked.

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

TRANSLATE PLC PROGRAM
The PLC program of the TNC is translated automatically.

RELAY EXT. DC VOLTAGE MISSING	
I	Switch on the control voltage. The TNC checks the EMERGENCY OFF circuit.

MANUAL OPERATION	
TRAVERSE REFERENCE POINTS	
I	Move the axes over the reference marks in the displayed sequence: For each axis press the START key, or
X **Y**	Cross the reference points in any sequence: Press the machine axis direction button for each axis, until the reference point has been traversed.

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

The reference points need only be traversed if the machine axes are to be moved. If you intend only to write, edit or test programs, you can select the PROGRAMMING AND EDITING or TEST RUN modes of operation immediately after switching on the control voltage. The reference points can then be traversed later by pressing the PASS OVER REFERENCE soft key in the MANUAL mode of operation.

Traversing reference points with a tilted working plane

407

In a tilted coordinate system, the reference points are traversed by pressing the machine axis direction buttons. To enable this function, set TILT WORKING PLANE to ACTIVE in the MANUAL OPERATION mode (see page 2-26). The TNC then interpolates the tilted axes as soon as the corresponding axis direction buttons are pressed.

The NC START key is disabled; pressing this key will display an error message.

The angular values entered in the menu must correspond to the actual angle of the tilt axis.

1.4 Graphics and Status Displays

In the program run operating modes (except on TNC 407) and test run operating modes, the TNC provides the following three display modes:

- Plan view
- Projection in three planes
- 3D view

The display mode is selected with the soft keys.

On the TNC 415 B, TNC 425 and TNC 426, workpiece machining can also be graphically simulated in real time.

The TNC graphic depicts the workpiece as if it were being machined by a cylindrical end mill. If tool tables are used, a spherical cutter can also be depicted (see page 4-10).

The graphics window will not show the workpiece if

- the current program has no valid blank form definition
- no program is selected

With machine parameters MP7315 to MP7317 a graphic is generated even if no tool axis is defined or moved.

The graphics cannot show rotary axis movements (error message).

Graphics during program run

407

A graphical representation of a running program is not possible if the microprocessor of the TNC is already occupied with complicated machining tasks or if large areas are being machined.

Example:

Stepover milling of the entire blank form with a large tool.

The TNC interrupts the graphics and displays the text "ERROR" in the graphics window. The machining process is continued, however.

1.4   Graphics and Status Displays

Plan view

The depth of the workpiece surface is displayed according to the principle "the deeper, the darker."

The number of displayable depth levels can be selected with the soft keys:

• TEST RUN mode: 16 or 32
• PROGRAM RUN modes: 16 or 32

Plan view is the fastest of the three graphic display modes.
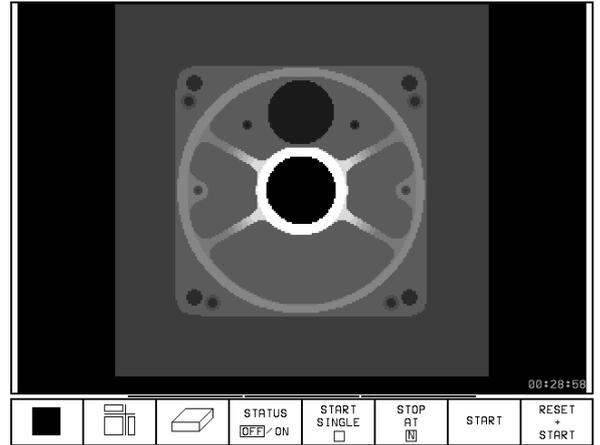
Fig. 1.21:   TNC graphics, plan view

| | Show 16 or 32 shades of depth. |
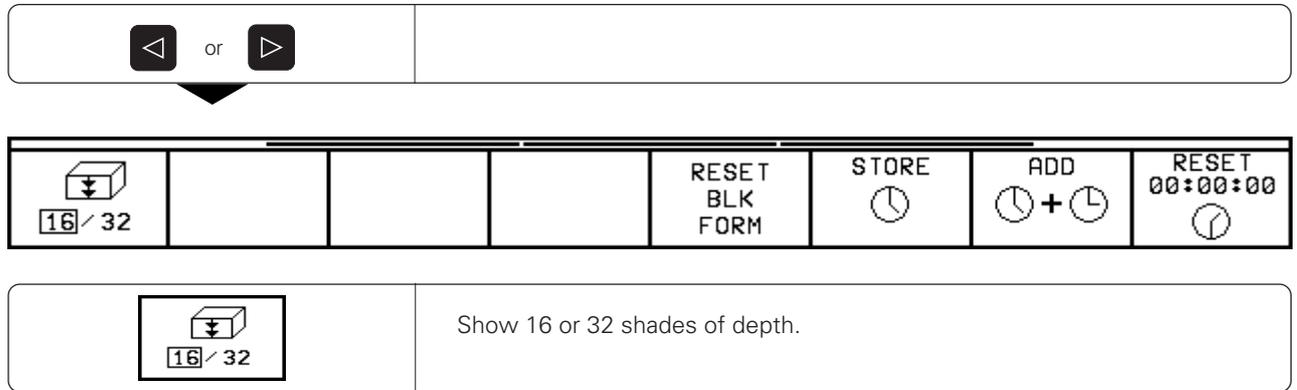
Projection in 3 planes

Similar to a workpiece drawing, the part is displayed with a plan view and two sectional planes. A symbol to the lower left indicates whether the display is in first angle or third angle projection according to ISO 6433 (selected with MP 7310).

Details can be isolated in this display mode for magnification (see page 1–25).
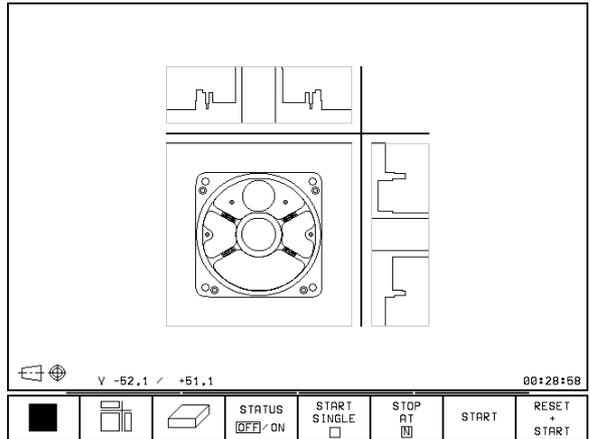
Fig. 1.22:   TNC graphics, projection in three planes

Shifting planes

The sectional planes can be shifted as desired. The positions of the sectional planes are visible during shifting.
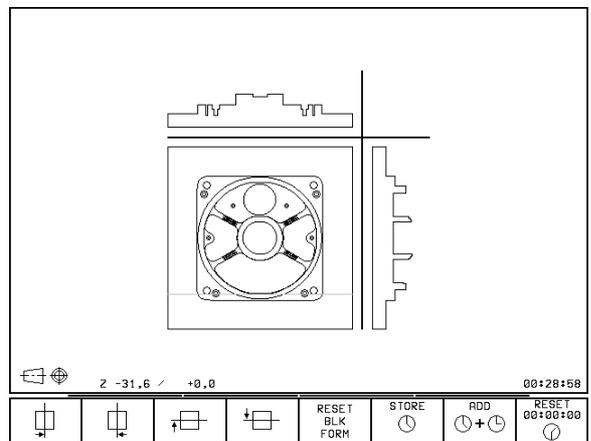
Fig. 1.23:   Shifting sectional planes

◁ or ▷	Shift the soft-key row.	

				RESET BLK FORM	STORE	ADD	RESET 00:00:00

or	Shift the vertical sectional plane to the right or left.	
or	Shift the horizontal sectional plane upwards or downwards.	

Cursor position during projection in 3 planes

The TNC shows the coordinates of the cursor position at the bottom of the graphics window. Only the coordinates of the working plane are shown.

This function is activated with machine parameter MP 7310.

Cursor position during detail magnification

During detail magnification, the TNC displays the coordinates of the axis that is currently being moved.

The coordinates describe the area determined for magnification. To the left of the slash is the smallest coordinate of the detail in the current axis, to the right is the largest.
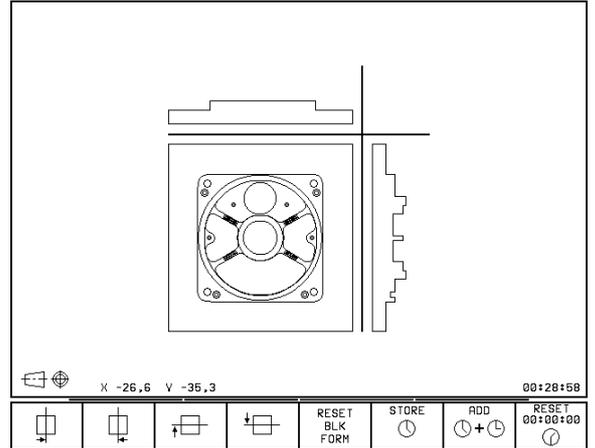
Fig. 1.24:   The coordinates of the cursor position are displayed to the lower left of the graphic

3D view
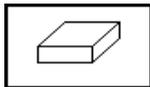
Here the workpiece is displayed in three dimensions, and can be rotated about the vertical axis.

The shape of the workpiece blank can be depicted by a frame overlay at the beginning of the graphic simulation.

In the TEST RUN mode of operation you can isolate details for magnification.
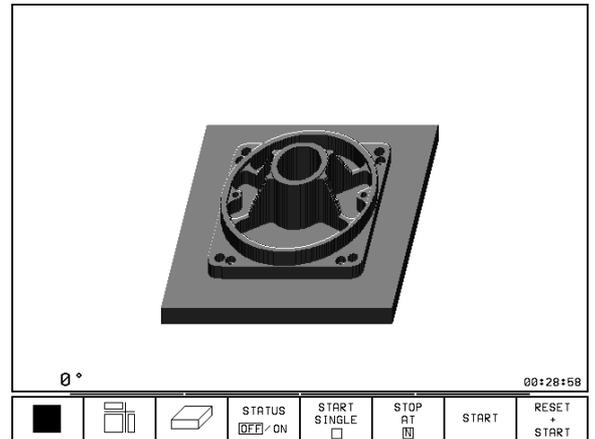
Fig. 1.25:   3D view

To rotate the 3D view:

| ◁ or ▷ | Shift the soft-key row. |

| [cube icon] [cube icon] | SHOW BLK-FORM | OMIT BLK-FORM | RESET BLK FORM | STORE ⊙ | ADD ⊙+⊙ | RESET 00:00:00 ⊘ |

| [cube icon] or [cube icon] | Rotate the workpiece in 27° steps about the vertical axis. |

The current angular attitude of the display is indicated at the lower left of the graphic.
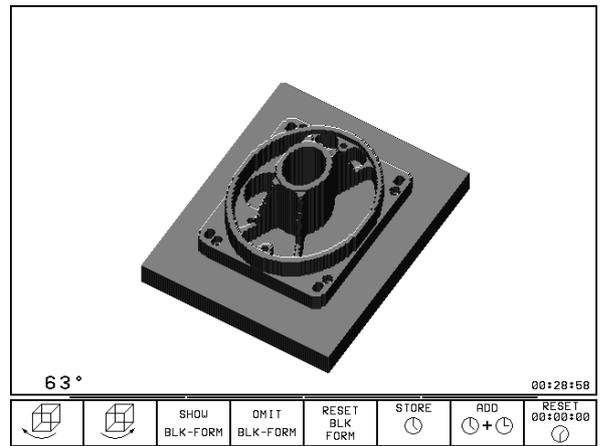
Fig. 1.26:   Rotated 3D view

To switch the frame overlay display on/off:

| SHOW BLK-FORM or OMIT BLK-FORM | Show or omit the frame overlay of the workpiece blank form. |

Magnifying details

You can magnify details in the TEST RUN mode of operation in the following display modes:

- projection in three planes
- 3D view

provided that the graphic simulation is stopped. A detail magnification is always effective in all three display modes.
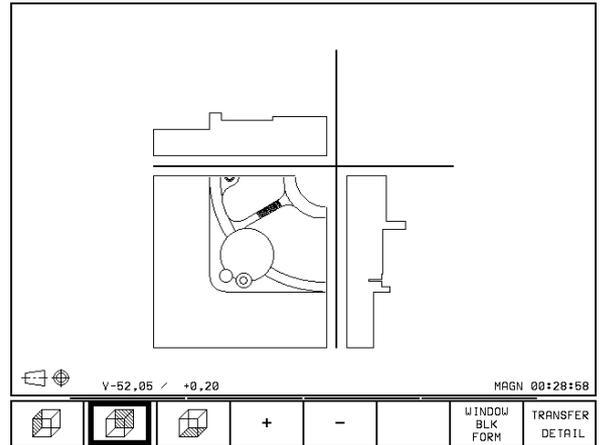
Fig. 1.27:   Magnifying a detail of a projection in three planes

To select detail magnification:

◁ or ▷	Shift the soft-key row.

			+	–		WINDOW BLK FORM	TRANSFER DETAIL

/	Select the left/right workpiece surface.
/	Select the front/back workpiece surface.
/	Select the top/bottom workpiece surface.

– or +	Shift sectional plane to reduce/magnify the blank form.

If desired TRANSFER DETAIL	Select the isolated detail.

Restart the test run or program run.
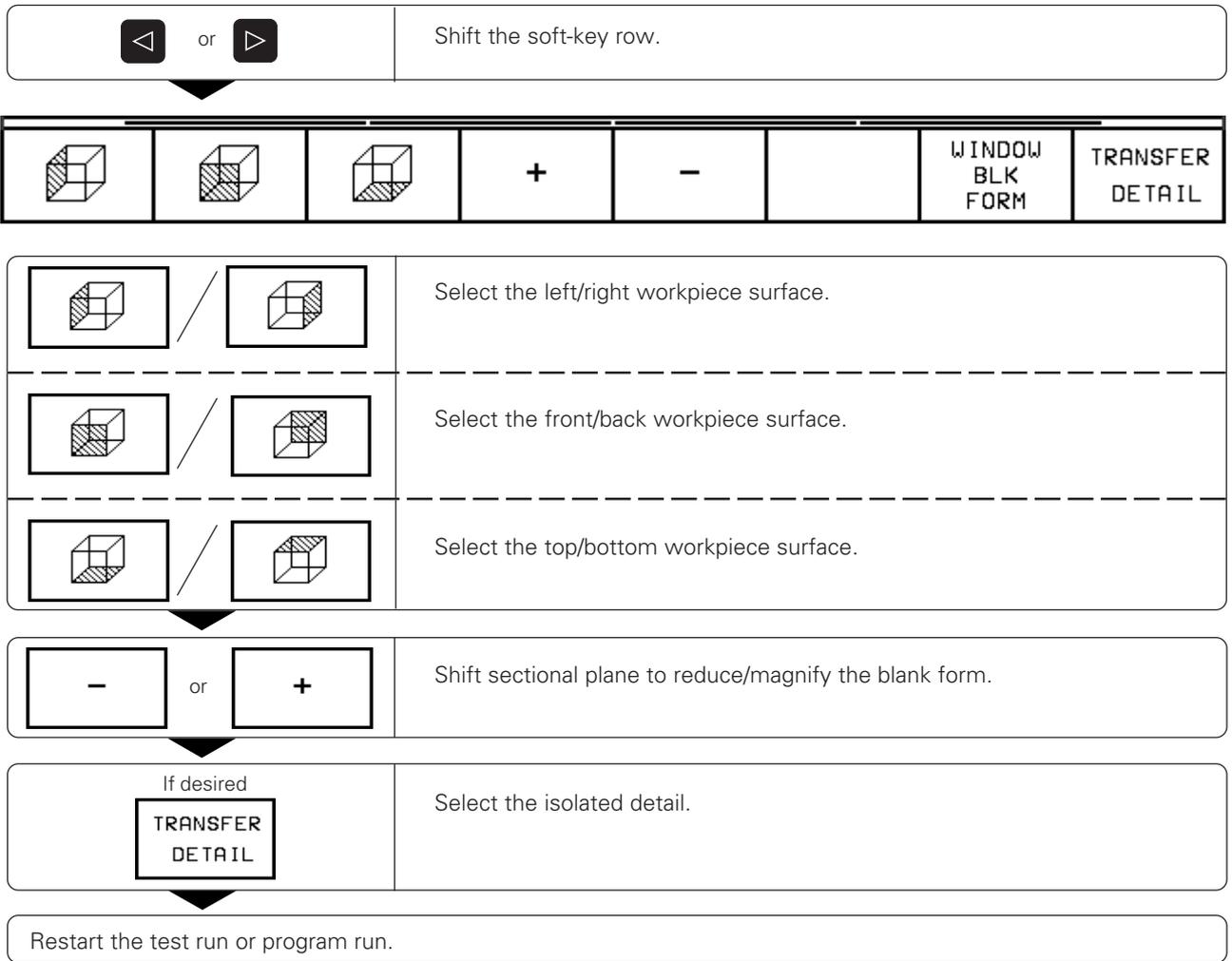
If a graphic display is magnified, this is indicated with MAGN at the lower right of the graphics window. If the detail is not magnified with TRANSFER DETAIL, you can make a test run of the shifted sectional planes.

> If the workpiece blank cannot be further enlarged or reduced, the TNC displays an error message in the graphics window. The error message disappears when the workpiece blank is enlarged or reduced.

Repeating graphic simulation

A part program can be graphically simulated as often as desired, either with the complete workpiece blank or with a detail of it.

Function	Soft key
Restore workpiece blank as it was last shown	RESET BLK FORM
Show the complete BLK FORM as it appeared before a detail was magnified via TRANSFER DETAIL	WINDOW BLK FORM

> ☞ The WINDOW BLK FORM soft key will return the blank form to its original shape and size, even if a detail has been isolated and not yet magnified with TRANSFER DETAIL.

Measuring the machining time

At the lower right of the graphics window the TNC shows the calculated machining time in

hours : minutes : seconds
(maximum 99 : 59 : 59)

- Program run:
  The clock counts and displays the time from program start to program end. The clock stops whenever machining is interrupted.

- Test run:
  The clock shows the time which the TNC calculates for the duration of tool movements.


```
PROGRAM RUN   TEST RUN
SINGLE BLOCK

%3815 G71 *
N10 D00 Q1 P01 +0 *
N20 D00 Q2 P01 +0 *
N30 D00 Q3 P01 -40 *
N35 D00 Q6 P01 +40 *
N36 D00 Q16 P01 +10 *
N40 D00 Q7 P01 -90 *
N50 D00 Q17 P01 +90 *
N60 D00 Q8 P01 +0 *
N70 D00 Q18 P01 +90 *
N80 D00 Q9 P01 +0 *
N90 D00 Q10 P01 +50 *
N100 D00 Q12 P01 +0 *
N110 *
N120 D00 Q20 P01 +500 *      63°        MAGN    00:43:31
```
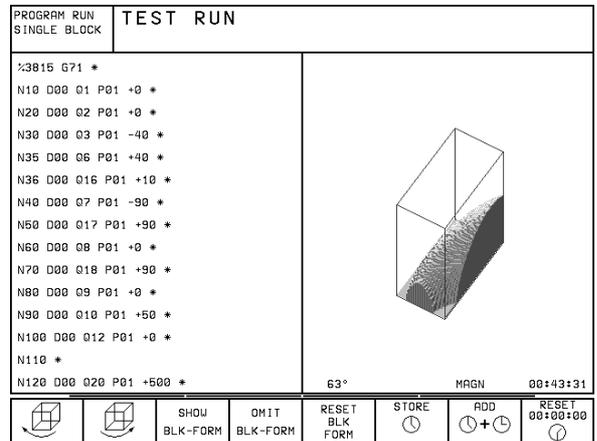
Fig. 1.28:   The calculated machining time is shown at the lower right of the workpiece graphic

To activate the stopwatch function:

◁ or ▷   Press the shift keys until the soft-key row with the stopwatch functions appears.

					STORE	ADD	RESET 00:00:00

> ☞ The soft keys available to the left of the stopwatch function depend on the selected display mode.

Stopwatch functions	Soft key
Store displayed time	STORE ⏱
Show the sum of the stored time and the displayed time	ADD ⏱+⏱
Clear displayed time	RESET 00:00:00 ⏱

Status displays

During a program run mode of operation the status display contains the current coordinates and the following information:

- Type of position display (ACTL, NOML, ...)
- Number of the current tool T
- Tool axis
- Spindle speed S
- Feed rate F
- Active M functions
- "Control in operation" symbol: ✳
- "Axis is locked" symbol: ⊶
- Axis can be moved with the handwheel: ⊘
- Axes are moving in a tilted working plane: ⬢
- Axes are moving under a basic rotation: ⬡

```
PROGRAM RUN / SINGLE BLOCK          TEST RUN

%3805 G71 *
N10 G30 G17 X+0 Y+0 Z-40 *
N20 G31 G90 X+100 Y+100 Z+0 *
N30 G99 T1 L+0 R+7,5 *
N40 T1 G17 S1500 *
N50 G00 G40 G90 Z+50 *
N60 G75 P01 -2 P02 -20 P03 -10 P04 100
    P05 X+80 P06 Y+50 P07 500 *
N70 X+50 Y+50 M3 *
                                        00:00:00
ACTL.      X    +132,6870    Y    +12,5600
           Z    +160,2560    B    +30,0000
           C     +90,0000
T                            F 0         M 5/9
```
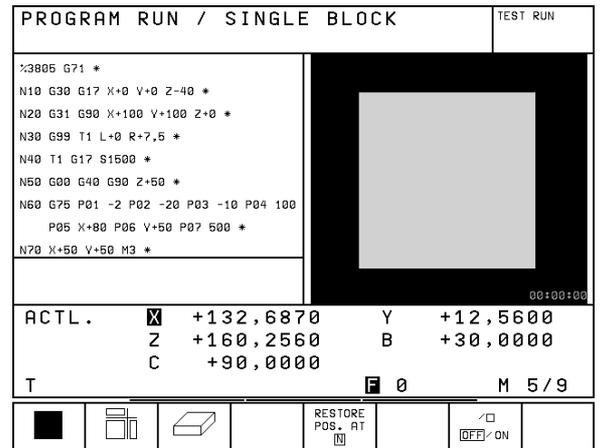
Fig. 1.29:   Status display in a program run mode of operation

Additional status displays

The additional status displays contain further information on the program run.

To select additional status displays:

STATUS OFF/ON	Set the STATUS soft key to ON.

◁ or ▷	Shift the soft-key row.

| STATUS PGM | STATUS POS. | STATUS TOOL | STATUS COORD. TRANSF. | STATUS TOOL PROBE | STORE ⏱ | ADD ⏱+⏱ | RESET 00:00:00 ⏱ |
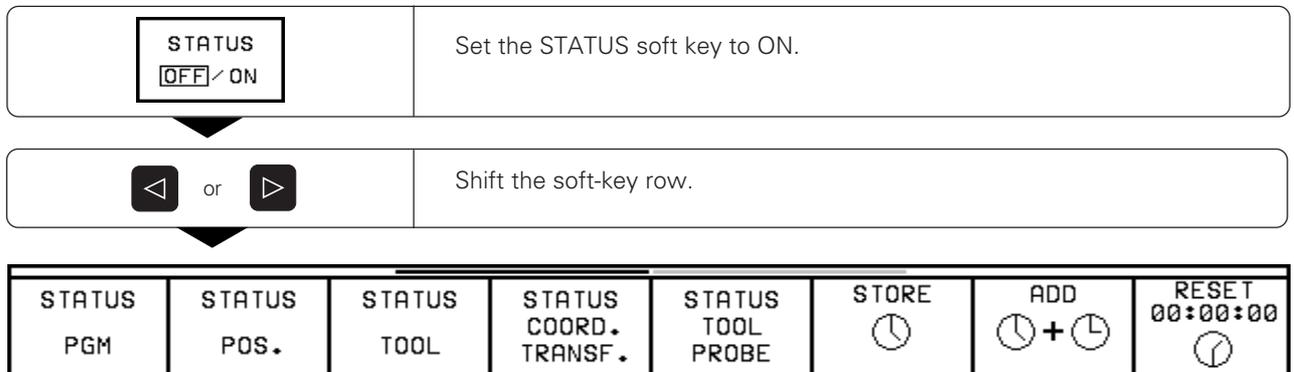
Additional status display	Soft key
General program information	STATUS PGM
Positions and coordinates	STATUS POS.
Tool information	STATUS TOOL
Coordinate transformations	STATUS COORD. TRANSF.
Tool measurement	STATUS TOOL PROBE

General program information

Name of main program
Active programs
Cycle definition
Dwell time counter
Machining time
Circle center CC (pole)

Positions and coordinates

Type of position display
Coordinates of the axes
Tilt angle of the working plane
Display of a basic rotation

Tool information

```
MANUAL      TEST RUN
OPERATION

N30 D01 Q53 P01 +Q3 P02 +Q23 *         TOOL      T 1 SCHRUPP1
N40 D01 Q56 P01 +Q6 P02 -Q108 *
N50 D00 Q58 P01 +Q8 *                  Z            L   -25.6340
N60 D04 Q72 P01 +Q12 P02 +Q26 *                     R   +12.5000
                                                    R2   +0.5000
N70 D03 Q72 P01 +Q72 P02 +Q29 *
N80 D02 Q77 P01 +Q17 P02 +Q7 *                 DL      DR      DR2
N90 D04 Q77 P01 +Q77 P02 +Q27 *        TAB  +0.2500  +0.1000
                                       PGM
N100 D03 Q77 P01 +Q77 P02 +Q29 *
N110 D02 Q78 P01 +Q18 P02 +Q8 *        ⊙   CUR.TIME  TIME1    TIME2
                                            04:13    07:30    06:40
N120 D04 Q78 P01 +Q78 P02 +Q26 *
N130 D03 Q78 P01 +Q78 P02 +Q29 *       TOOL CALL   1 SCHRUPP1
N140 D02 Q76 P01 +Q16 P02 +Q6 *        RT  ⟷     12
N150 D04 Q76 P01 +Q76 P02 +Q26 *
N160 D03 Q76 P01 +Q76 P02 +Q29 *
N170 G54 X+Q1 Y+Q2 Z+Q53 *

STATUS   STATUS   STATUS   STATUS
PGM      POS.     TOOL     COORD.
                           TRANSF.
```

T: Tool name and number
RT: Name and number of a replacement tool

Tool axis

Tool length and radii

Oversizes (delta values)

Tool life, maximum tool life and maximum tool life for TOOL CALL

Display of the programmed tool and the (next) replacement tool

Coordinate transformations

```
PROGRAM RUN   TEST RUN
FULL SEQUENCE

N6 G28 X *                          PROGRAMS      3813
N7 G73 G90 H+13.66 *
N8 G72 F1.01115 *                    DATUM SHIFT          ROTATION
N9 G38 *                               X  +12.5000         +346.4400
N10 G00 G40 G90 Z+Q25 M3 *             Y  +12.5000
                                       Z   -2.0000      MIRROR IMAGE
N20 D00 Q52 P01 +Q *                                       X
N30 D01 Q53 P01 +Q3 P02 +Q23 *
N40 D01 Q56 P01 +Q6 P02 -Q108 *            SCALING
N50 D00 Q58 P01 +Q8 *                 X  +0.0000   1.011150
N60 D04 Q72 P01 +Q12 P02 +Q26 *       Y  +0.0000   1.011150
N70 D03 Q72 P01 +Q72 P02 +Q29 *       Z  +0.0000   1.011150
N80 D02 Q77 P01 +Q17 P02 +Q7 *
N90 D04 Q77 P01 +Q77 P02 +Q27 *
N100 D03 Q77 P01 +Q77 P02 +Q29 *
N110 D02 Q78 P01 +Q18 P02 +Q8 *

STATUS   STATUS   STATUS   STATUS
PGM      POS.     TOOL     COORD.
                           TRANSF.
```

Main program name

Coordinates of the datum shift

Angle of basic rotation

Mirrored axis

Scaling factor(s)

Scaling datum
(conversational programming only)

Tool measurement (only available with conversational programming)

```
PROGRAM RUN / FULL SEQUENCE       TEST RUN

2  TCH PROBE 32.0 CAL. TOOL RADIUS     TOOL     T 1
3  TCH PROBE 32.1 CHECK: 0
4  TCH PROBE 32.2 HEIGHT: +30        R    MIN 2   +8.4171
5  TCH PROBE 32.3 PROBING THE EDGES: 1      MAX 1   +8.7554
6  END PGM MESS MM                         DYN     +8.8964

                                       1   +8.7554 *
                                       2   +8.4171 *
                                       3   +8.7293 *
                                       4   +8.7464 *

ACTL.    X     +0.9608      Y     +1.0674
         Z     +1.0839      B     +1.0849
         C     +2.0487
T 1    Z                    F              M 5/9

PAGE    PAGE    BEGIN   END     RESTORE           ⁄▢      TOOL
 ⇧       ⇩      TEXT    TEXT    POS. AT          OFF/ON   TABLE
                                 N
```

Number of the tool to be measured

Measured MIN and MAX values of the single cutting edges and the result of measuring the rotating tool

Display whether the tool radius or the tool length is being measured

Cutting edge number with the corresponding measured value. If the measured value is followed by an asterisk, the allowable tolerance defined in the tool table was exceeded.

1.5   File Management on the TNC 426

☞ The functions described in this chapter are valid for the hard disk of the TNC and for the FE1 interface mode (see page 10-4). If you wish to use these functions with a personal computer, you will need the HEIDENHAIN data transfer software TNC.EXE.

Programs, texts and tables are written as **files** and stored on the TNC's hard disk.

File identification:

PROG15	.I

File name   File type
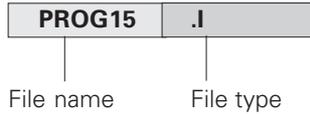
To open a new file you must enter a file name consisting of up to eight characters (letters and numbers).

You can manage any number of files on the TNC's hard disk. Their total size, however, must not exceed 170 M byte.

To ensure that you can easily find your files, we recommend that you organize your hard disk into **directories**. Directories work like drawers in a filing cabinet: They enable you to save your files in groups according to some system. You could set up your directories according to job number, for example. The name of a directory can have up to eight characters (letters and numbers).

If you divide a directory up into further directories, these subordinate directories are called **subdirectories**. The TNC shows subdirectories at the right of and below their parent directories (see page 1-31).

Files in the TNC	Type
Programs	
• in HEIDENHAIN plain language dialog	.H
• in ISO format	.I
Tables for	
• Tools	.T
• Pallets	.P
• Datums	.D
• Points (digitizing range for measuring touch probe)	.PNT
Texts as	
• ASCII files	.A

Fig. 1.35:   Overview of file types in the TNC 426
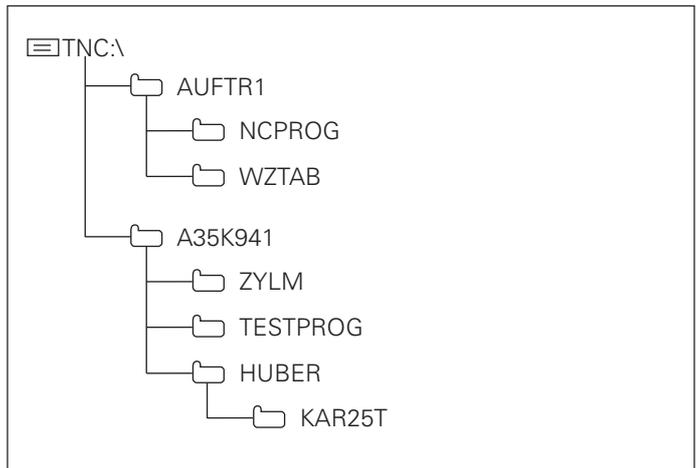
Fig. 1.36:   Directories on the hard disk of the TNC

Data security

We recommend to save newly written programs and files on a PC at regular intervals. You can do this with the cost-free backup program TNCBACK.EXE from HEIDENHAIN.

In addition, you need a floppy disk on which all machine-specific data, such as PLC program, machine parameters, etc., are stored. Please contact your machine manufacturer for more information on both the backup program and the floppy disk.

Saving the entire hard disk (170 M bytes) may take several hours. In this case, it is a good idea to save the data outside of work hours, e.g. during the night.

Calling the file manager

Press the PGM MGT key to call the file manager. The screen will then look something like this:

```
MANUAL          PROGRAMMING AND EDITING
OPERATION       FILE NAME =21070.H                    ──── Active path or
                                                           file name
 LWL:\           TNC:\TEST\HDH\KONTUR\ALT\*.H
 RS232:\                                              ──── File information
 RS422:\          FILE NAME      BYTE   STATUS DATE       TIME
 TNC:\ <--        21000      .H    702       28-02-2006 08:26:50
                  21010      .H   1006       28-02-2006 08:26:50
                  21020      .H   1184       28-02-2006 08:26:52
   TEST           21030      .H    454       28-02-2006 08:26:52
    HDH           21040      .H   1144       28-02-2006 08:26:54
     KONTUR       21041      .H   1136       28-02-2006 08:26:54
      ALT <--     21050      .H   1026       28-02-2006 08:26:56
                  21060      .H    872       28-02-2006 08:26:56
                  21070      .H    872       28-02-2006 08:26:56
                  21080      .H   1182       28-02-2006 08:26:58
                  21090      .H   1078       28-02-2006 08:26:58
                  113 FILE(S)   31516 KBYTE VACANT

  PAGE   PAGE   SELECT   COPY    SELECT   WINDOW            END
   ⇑      ⇓              ABC⇒XYZ  TYPE    ≡≡ ≡
```

With the WINDOW soft key you can change the screen layout (see page 1-34). Whenever you press the PGM MGT key, the TNC shows the screen layout that you last selected.

Drives:

RS232:   RS232 interface

RS422:   RS422 interface

TNC:\ <--   TNC disk drive

The active drive is shown in a different color. The drive symbol depends on the selected interface mode (see page 10-4).

Directories:

☐ NK

☐ KLART <--

☐ 3D

☐ BOHREN

☐ KDNET

The TNC shows a subdirectory at the right of and below its parent directory. The active directory is depicted in a different color and is indicated by an open file symbol.

File information:

• **FILE NAME:**	Files stored in the active directory
• **BYTES:**	Size of the file in bytes
• **STATUS:**	The following letters may appear in the STATUS column:
E:	File is selected in the PROGRAMMING AND EDITING operating mode
S:	File is selected in the TEST RUN operating mode
M:	File is selected in a program run operating mode
P:	File is protected against editing and erasure
IN:	File contains inch dimensions
W:	File was incompletely transferred to external storage and cannot be run
• **DATE:**	Date the file was last changed
• **TIME:**	Time the file was last changed

To select the drive:

➤ Press the leftward arrow key to move the highlight bar to the active directory (left in the screen).
➤ Press the upward arrow key to move the highlight bar to the desired drive.
➤ Once the highlight bar is on desired drive, confirm your selection with ENT.

To create a new directory:

➤ Select the drive.
➤ Press the downward arrow key to move the highlight to the directory under which you wish to open a new directory.
➤ Enter the name of the new directory (up to eight characters) and confirm with ENT.
➤ Answer the TNC dialog question with the YES soft key if you wish to create a new directory, or abort with NO.

To select the directory:

➤ Select the drive.
➤ Press the downward arrow key to move the highlight to the desired directory. In the right screen window, the TNC automatically displays the files stored in this directory.

or

➤ Press the leftward arrow key to move the highlight to the active directory (left in the screen).
➤ Enter the path directly with the ASCII keyboard and confirm with ENT.

1.5   File Management on the TNC 426

To select the file:

➤ Select the drive.
➤ Select the directory.
➤ Press the rightward arrow key to move the highlight to the file directory.
➤ Press the SELECT TYPE soft key to choose the file type (see page 1-36).
➤ Press the downward or upward arrow key to move the highlight to the desired file.
➤ Once the highlight is on the desired file, confirm your selection with ENT.

or

➤ Press the LAST FILES soft key. The TNC then displays the last ten files you selected.
➤ Press the downward or upward arrow key to move the highlight to the desired file.
➤ Once the highlight is on the desired file, confirm your selection with ENT.

or

➤ Select the drive.
➤ Select the directory.
➤ Enter the file name directly with the ASCII keyboard.

Functions for selecting files	Key / Soft key
Move the highlight bar upward to the desired file	⬆
Move the highlight bar downward to the desired file	⬇
Move pagewise up through the file directory	PAGE ⇧
Move pagewise down through the file directory	PAGE ⇩
Select the highlighted file	SELECT

1.5   File Management on the TNC 426

With the WINDOW soft key you can change the screen layout to show the contents of two different directories, one in each half of the screen. Use this setting for copying files between directories and for importing or exporting files. Press the WINDOW soft key again if you wish to return to the other layout.

File name

Active path
File information

Active path
File information

File display:
Files stored in the active directory

File display:
Files stored in the active directory

```
MANUAL            PROGRAMMING  AND  EDITING
OPERATION         FILE  NAME  =

TNC:\TEST\HDH\KONTUR\ALT\*.H        TNC:\*.H
 FILE NAME        BYTE  STATUS       FILE NAME        BYTE  STATUS
21000      .H      702              #1         .H       54
21010      .H     1006              $MDI       .H      152
21020      .H     1184              0          .H      174
21030      .H      454              00         .H      154
21040      .H     1144              000        .H     6526
21041      .H     1136              0000X      .H       30
21050      .H     1026              0020202    .H       12
21060      .H      872              00X        .H       12
21070      .H      872              01         .H      260
21080      .H     1182              01234      .H     6524
21090      .H     1078              012345     .H     6524
113 FILE(S)  31516 KBYTE VACANT     195 FILE(S)  31516 KBYTE VACANT

  PAGE    PAGE   SELECT   COPY    SELECT  WINDOW
   ⇑       ⇓      ⤏      ABC⇒XYZ   TYPE    ⊟⊟ ═    PATH   END
```

To select the directory:

➤ Press the leftward or rightward arrow key to switch to the window in which you wish to select a new directory.

➤ Press the PATH soft key to switch to the path display. The TNC then shows the drives (active drive is depicted in a different color) and the active directory (also depicted in a different color).

➤ If you wish to change drives, press the upward arrow key to move the highlight to the new drive.

➤ Once the highlight is on the desired drive, confirm your selection with ENT. The TNC automatically returns to the file display. The window then shows the files stored in the uppermost directory of the selected drive.

➤ If you wish to select a subdirectory, press the PATH soft key to return to the path display.

➤ Press the downward arrow key to move the highlight to the desired directory.

➤ Once the highlight is on the desired directory, confirm your selection with ENT. The TNC then automatically switches back to the file window containing the files stored in the selected directory.

Functions for file management

The file management functions are selected by soft key after pressing PMG MGT in the PROGRAMMING AND EDITING mode of operation. The following functions are available:

Function	Soft key
Select a file type	SELECT TYPE
Copy a file (and convert)	COPY ABC⇒XYZ
Copy a directory	COPY DIR
Erase a file or directory	DELETE
Rename a file	RENAME ABC = XYZ
Tag files	TAG
Read the tree structure of an external device and show it on the TNC screen	SHOW TREE
Display the last ten files that were selected	LAST FILES
Protect a file	PROTECT
Cancel file protection	UNPROTECT
Convert FK program	CONVERT FK->H
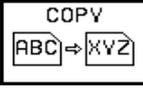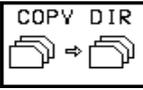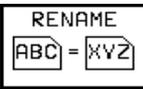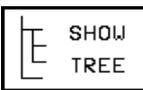
Selecting file types

PGM MGT	Call the file manager.
SELECT TYPE	List the file types.

SHOW ALL	SHOW .H	SHOW .T	SHOW .I	SHOW .P	SHOW .D	SHOW .A	END

SHOW ALL	Show all files, or ...
SHOW .H	show only one type, for example HEIDENHAIN conversational programs (file type .H).
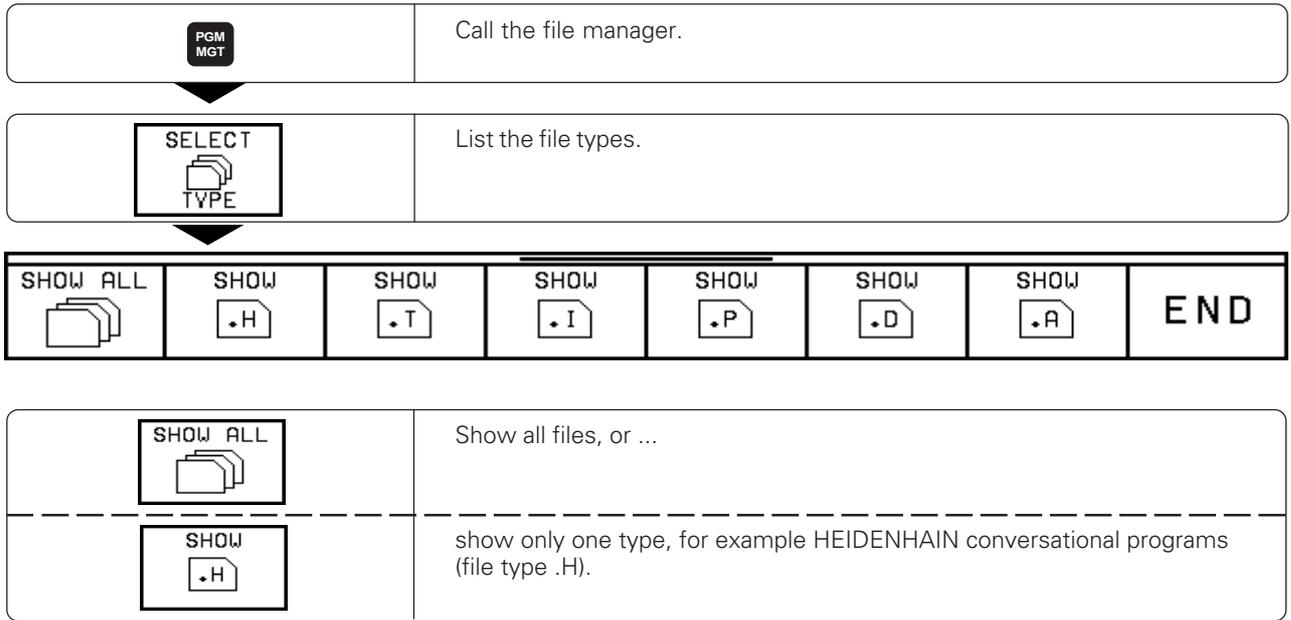
To copy individual files:

You must be in the PROGRAMMING AND EDITING mode of operation.

➤ Call the file manager with PGM MGT.
➤ Select the directory containing the file you wish to copy. Move the highlight to the desired file.
➤ Press the COPY soft key.
➤ Type the new file name into the highlight in the screen headline.
➤ Press the ENT key or the EXECUTE soft key to copy the file into the active directory. The original file is retained.
➤ Close the file manager with END.

• If you are copying tables, you can overwrite individual lines or columns in the target table with the REPLACE FIELDS soft key. Prerequisites:
  – The target table must exist.
  – The file to be copied must only contain the columns (or lines) you want to replace.
• You can also copy an entire directory together with its subdirectories. Simply place the highlight on the directory you wish to copy and press the COPY DIR soft key. Then proceed as described above in the section "To copy individual files".

To copy several files into another directory:

You must be in the PROGRAMMING AND EDITING mode of operation.
To select the functions for copying several files, press the TAG soft key:

Functions for tagging/copying files	Soft key
Tag one file	TAG FILE
Tag all files in the directory	TAG ALL FILES
Untag one file	UNTAG FILE
Untag all files in the directory	UNTAG ALL FILES
Copy the tagged files	COPY TAG
When overwriting several files, confirm each file separately	CONFIRM
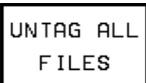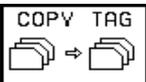
➤ Call the file manager with PGM MGT.
➤ Arrange the screen layout with the WINDOW soft key to show file names in both halves of the screen.
➤ In the right screen half, use the PATH soft key to select the directory into which you wish to copy the files.
➤ Move the highlight to the left screen half.
➤ In the left screen half, use the PATH soft key to select the directory containing the files that you want to copy.
➤ Move the highlight to the first file you want to copy.
➤ Shift the soft-key row.
➤ Select the file tagging function with the TAG soft key.
➤ Press the TAG FILE soft key. The TNC tags the highlighted file with an arrow at its left and shows it in a different color.
➤ Move the highlight to the next file you want to copy. Tag the file with TAG FILE. Tag all files you want to copy in this way.
➤ Press the COPY TAG soft key and confirm with ENT. Die TNC copies the tagged files into the active directory in the right screen half. The original files are retained.
➤ Close the file manager with END.

- If the target directory contains files with the same file names as the files to be copied, the TNC will ask you whether you want to overwrite these files. Press the YES soft key to overwrite all files, or press the CONFIRM soft key to confirm each file separately before overwriting it. If you want to overwrite a protected file, you can suspend file protection during the copying process.
- When you are converting files with the TAG function, you can also use *wildcards* (∗). By copying into ∗.A, for example, you can convert all tagged files into ASCII text files.
- If you press the COPY TAG soft key with the screen layout showing drives/directories to the left and file names to the right, the TNC asks you for a DESTINATION DIRECTORY. Enter the complete path name, including the drive.

To erase a file:

> ➤ Call the file manager with PGM MGT.
> ➤ Select the directory containing the file you wish to erase. Move the highlight to the desired file.
> ➤ Shift the soft-key row.
> ➤ Press the DELETE soft key.
> ➤ Press the YES soft key to delete the file, or abort the deleting function with the NO soft key.
> ➤ Close the file manager with END.

> You can also erase directories. Simply move the highlight to a directory instead of a file and proceed as described above. Delete all files and subdirectories stored in this directory before erasing the actual directory.

To rename a file:

> ➤ Call the file manager with PGM MGT.
> ➤ Select the directory containing the file you wish to rename. Move the highlight to the desired file.
> ➤ Shift the soft-key row.
> ➤ Press the RENAME soft key and enter the new file name.
> ➤ Press the ENT key or the EXECUTE soft key to rename the file. The original file name is erased.
> ➤ Close the file manager with END.

To protect a file:

> ➤ Call the file manager with PGM MGT.
> ➤ Select the directory containing the file you wish to protect. Move the highlight to the desired file.
> ➤ Shift the soft-key row.
> ➤ Press the MORE FUNCTIONS soft key.
> ➤ Press the PROTECT soft key. The file now has status P and cannot be accidentally changed or erased.
> ➤ Close the file manager with END.

To cancel file protection:

> ➤ Call the file manager with PGM MGT.
> ➤ Select the directory containing the file whose protection you wish to remove. Move the highlight to the desired file.
> ➤ Shift the soft-key row.
> ➤ Press the MORE FUNCTIONS soft key.
> ➤ Press the UNPROTECT soft key.
> ➤ Type the code number 86357 and confirm with ENT. File protection is canceled, the file no longer has status P.
> ➤ Close the file manager with END.

To convert a file:

Two soft keys are provided for converting files:

Functions for converting files	Soft key
FK program to HEIDENHAIN conversational format	CONVERT FK->H
HEIDENHAIN conversational format to OEM cycle	CONVERT H->CYC

To copy a file type into ASCII format (.A), use the COPY function (see page 1-36). Enter the file type .A for the destination file.

You can also convert an ASCII file into another format. To convert to ISO format, for example, enter the file type .I for the destination file.

Example: Converting an FK program into HEIDENHAIN conversational format

➤ Call the file manager with PGM MGT.
➤ Select the directory containing the file you wish to convert. Move the highlight to the desired file.
➤ Shift the soft-key row.
➤ Press the MORE FUNCTIONS soft key.
➤ Press the CONVERT FK –> H soft key.
➤ Type the new file name into the highlight in the screen headline and confirm with ENT.
➤ Close the file manager with END.

1.6   File Management on the TNC 425, TNC 415 B and TNC 407

Programs, texts and tables are written as files and stored in the TNC.

File identification:

PROG15	.I
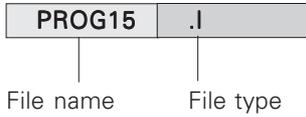
File name   File type

To open a new file you must enter a file name consisting of from one to 16 characters (letters and numbers), depending on MP7222.

The file types are listed in the table at right.

Files in the TNC	Type
Programs	
• in HEIDENHAIN plain language dialog	.H
• in ISO format	.I
Tables for	
• Tools	.T
• Pallets	.P
• Datums	.D
• Points (digitizing range for measuring touch probe)	.PNT
Texts as	
• ASCII files	.A

Fig. 1.39:   TNC file types

File directory

The TNC can store up to 100 files at one time. You can call up a directory of these programs by pressing the PGM NAME key. To delete one or more programs, press the CL PGM key.

The file directory contains the following information:

• File name
• File type
• File size in bytes (=characters)
• File status

Further information is shown at the top of the screen:

• Selected file storage
  - TNC memory
  - External storage over RS-232 interface
  - External storage over RS-422

• Interface mode (e.g., FE1, EXT1 for external storage)

• File type (e.g., ✳ .H if only HEIDENHAIN dialog programs are shown)

Example:

RS 422/EXT1: ✳ .T is displayed. This means that only those files are shown that have the extension .T and are located in an external storage device (e.g. a PC) that is connected to the TNC over the RS-422 interface (see also Chapter 10).

A soft key calls the file directory of an external data storage medium. The screen is then divided into two columns.

Select the file directory:

Task	Mode of operation	Call file directory with
Create new files	◈	PGM NAME
Edit files	◈	PGM NAME
Erase files	◈	CL PGM
Test files	⊡	PGM NAME
Execute files	⊡ ⊡	PGM NAME

Fig. 1.40:   File management functions

MANUAL OPERATION	DATUM TABLE EDITING

FILE NAME = PAL2   .P

TNC:

FILE NAME		BYTE	STATUS
76134	.H	1788	
79116	.H	1650	
79152	.H	1482	
79153	.H	1100	
FRESADOR	.H	462	
ERFG	.I	74	
PAL1	.P	756	
PAL2	.P	756	
DD25LP	.D	462	
SK50	.D	462	E
11Y	.A	696	
JKL	.A	1008	

53 FILE(S) 155904 BYTES VACANT

| PAGE ⇑ | PAGE ⇓ | SELECT | COPY ABC⇒XYZ | | SELECT TYPE | WINDOW =|= ▤ | END |

Fig. 1.41:   Files are sorted alphabetically and according to type

WINDOW =|= ▤	Show the file directory in one or two columns. The selected layout is shown in the soft key.

1.6   File Management on the TNC 425, TNC 415 B and TNC 407

File status

The letters in the STATUS column give the following information about the files:

E: File is selected in the PROGRAMMING AND EDITING
        operating mode
S: File is selected in the TEST RUN operating mode
M: File is selected in a program run operating mode
P: File is protected against editing and erasure
IN: File contains inch dimensions
W: File has been transferred to external storage and cannot be run

Selecting a file

You must be in the PROGRAMMING AND EDITING mode of operation.

➤ Call the file manager with PGM NAME.
➤ Display the file type soft-key row with the SELECT TYPE soft key. Then press the soft key for the desired file type, for example SHOW .I for ISO programs.
➤ Use the arrow keys to move the highlight bar to the desired file.
➤ Confirm your selection with the SELECT soft key or with ENT.
➤ Close the file manager with END.

Functions for selecting files	Key/ Soft key
Move the highlight bar upward to the desired file	↑
Move the highlight bar downward to the desired file	↓
Move pagewise up through the file directory	PAGE ⇧
Move pagewise down through the file directory	PAGE ⇩
Select the highlighted file	SELECT

1.6   File Management on the TNC 425, TNC 415 B and TNC 407
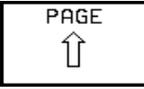
To copy a file:

➤ Call the file manager with PGM NAME.
➤ Move the highlight to the file you wish to copy.
➤ Press the COPY soft key.
➤ Type the new file name into the highlight in the screen headline.
➤ Press ENT to copy the file. The original file is retained.
➤ Close the file manager with END.

To erase a file:

➤ Press CL PGM.
➤ Move the highlight to the file you wish to delete.
➤ Press the DELETE soft key to erase the highlighted file.
➤ Close the file manager with END.

To rename a file:

➤ Call the file manager with PGM NAME.
➤ Move the highlight to the file you wish to rename.
➤ Shift the soft-key row.
➤ Press the RENAME soft key and type the new file name into the
   highlight in the screen headline.
➤ Press ENT to rename the file. The original file name is erased.
➤ Close the file manager with END.

To protect a file:

➤ Call the file manager with PGM NAME.
➤ Move the highlight to the file you wish to protect.
➤ Shift the soft-key row.
➤ Press the PROTECT soft key. The file now has status P and cannot be
   accidentally changed or erased.
➤ Close the file manager with END.

To cancel file protection:

➤ Call the file manager with PGM MGT.
➤ Move the highlight to the file whose protection you wish to remove.
➤ Shift the soft-key row.
➤ Press the UNPROTECT soft key.
➤ Type the code number 86357 and confirm with ENT. File protection is
   canceled, the file no longer has status P.
➤ Close the file manager with END.

To convert a file:

Text files (type .A) can be converted to any other type. Other types of files can only be converted into ASCII text files. They can then be edited with the alphanumeric keyboard.

Part programs that were created with FK free contour programming can also be converted to HEIDENHAIN conversational programs.

➤ Call the file manager with PGM NAME.
➤ Move the highlight to the file you wish to convert.
➤ Shift the soft-key row.
➤ Press the CONVERT soft key
➤ Press the CONVERT FK –> H soft key.
➤ Type the new file name into the highlight in the screen headline and confirm with ENT.
➤ Close the file manager with END.

File management for files on external data media

You can erase and protect files stored on the FE 401B floppy disk unit from HEIDENHAIN. You can also format a floppy disk from the TNC. To do this you must first select the PROGRAMMING END EDITING mode of operation.

To erase a file on the FE 401B:

➤ Press the EXT key. The TNC displays the files stored in the TNC memory in the left screen half, and the files stored on the FE 401 in the right screen half.
➤ Move the highlight to the right screen half.
➤ Press the WINDOW soft key to select one-window mode.
➤ Move the highlight to the file you wish to delete.
➤ Press the DELETE soft key.
➤ Close the file manager with END.

To protect a file on the FE 401B:

➤ Press the EXT key. The TNC displays the files stored in the TNC memory in the left screen half, and the files stored on the FE 401 in the right screen half.
➤ Move the highlight to the right screen half.
➤ Press the WINDOW soft key to select one-window mode.
➤ Move the highlight to the file you wish to protect.
➤ Shift the soft-key row.
➤ Press the PROTECT soft key. The file now has status P and cannot be accidentally changed or erased.
➤ Close the file manager with END.

To cancel file protection on the FE 401B:

➤ Press the EXT key. The TNC displays the files stored in the TNC memory in the left screen half, and the files stored on the FE 401 in the right screen half.
➤ Move the highlight to the right screen half.
➤ Press the WINDOW soft key to select one-window mode.
➤ Move the highlight to the file whose protection you wish to remove.
➤ Shift the soft-key row.
➤ Press the UNPROTECT soft key.
➤ Type the code number 86357 and confirm with ENT. File protection is canceled, the file no longer has status P.
➤ Close the file manager with END.

1.6   File Management on the TNC 425, TNC 415 B and TNC 407

To format a floppy disk in the FE 401B:

➤ Press the EXT key. The TNC displays the files stored in the TNC memory in the left screen half, and the files stored on the FE 401 in the right screen half.
➤ Move the highlight to the right screen half.
➤ Press the WINDOW soft key to select one-window mode.
➤ Shift the soft-key row.
➤ Press the FMT soft key.
➤ Enter a name for the diskette and start formatting with ENT.
➤ Close the file manager with END.

To convert and transfer files:

➤ Press the EXT key.
➤ Move the highlight in the left screen half to the file you wish to convert and transfer.
➤ Shift the soft-key row.
➤ Press the CONVERT soft key
➤ Press the CONVERT –> .A soft key, for example, to convert the file into an ASCII text file and store it on the FE 401.
➤ Type the new file name into the highlight in the screen headline and confirm with ENT.
➤ Close the file manager with END.

2.1 Moving the Machine Axes

Traversing with the machine axis direction buttons

> Traversing with the machine axis direction buttons can vary depending on the individual machine tool. Your machine manual provides more information on this function.

MANUAL OPERATION

| e.g. (X) | The axis moves as long as the corresponding axis direction button is held down. |

You can move more than one axis at once in this way.

For continuous movement

MANUAL OPERATION

| e.g. (Y) (I) together | Press and hold the machine axis direction button, then press the machine START button. The axis continues to move after you release the keys. |
| (O) | To stop the axis, press the machine STOP button. |

You can move more than one axis at once in this way.

Traversing with an electronic handwheel

⊚ ▶	**ELECTRONIC HANDWHEEL**	
	INTERPOLATION FACTOR:	X = **3**

e.g. **3** **ENT**	Enter the interpolation factor (see table).	
e.g. **X**	Select the axis that you wish to move. For portable handwheels make the selection at the handwheel; for integral handwheels, at the TNC keyboard.	

Now move the selected axis with the electronic handwheel. If you are using the portable handwheel, first press the enabling switch (on side of handwheel).

Interpolation factor	Traverse in mm per revolution
0	20.000
1	10.000
2	5.000
3	2.500
4	1.250
5	0.625
6	0.312
7	0.156
8	0.078
9	0.039
10	0.019

Fig. 2.1:   Interpolation factors for handwheel speed   Fig. 2.2:   HR 330 electronic handwheel

> The smallest programmable interpolation factor depends on the specific machine tool. Your machine manual provides more detailed information on this subject.

Using the HR 330 electronic handwheel

The HR 330 portable handwheel has an enabling switch that is located on the side opposite to the star grip and the EMERGENCY STOP switch. You can only move the machine axes when the enabling switch is depressed.

> • As long as the handwheel is attached to the machine tool, the enabling switch is automatically depressed.
> • Attach the electronic handwheel to a steel surface with the mounting magnets such that it cannot be operated unintentionally.
> • Be sure not to press the axis direction keys unintentionally while the enabling switch is depressed when you remove the handwheel from its position.

Incremental jog positioning

With incremental jog positioning, a machine axis moves by a preset distance each time you press the corresponding machine axis direction button.
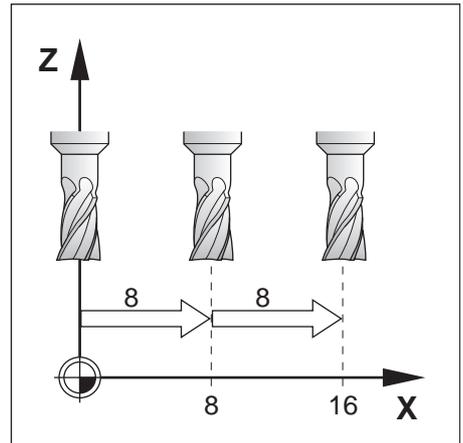
Fig. 2.3:   Incremental jog positioning in the X axis

ELECTRONIC HANDWHEEL	
INTERPOLATION FACTOR: X = 4	
I	Select incremental jog positioning with a key defined by your machine manufacturer (here, I).

ELECTRONIC HANDWHEEL		
JOG INCREMENT: 4 8		
e.g. 8 ENT	Enter the jog increment (here, 8 mm).	
e.g. X	Press the machine axis direction button as often as desired.	

- Incremental jog positioning can vary depending on the individual machine tool. Your machine manual provides more information on this function.
- The machine tool builder determines whether the interpolation factor for each axis is set at the keyboard or with a step switch.

Positioning with manual data input (MDI)

Machine axis movement can also be programmed in the $MDI file (see page 5-45).

Since the programmed movements are stored in memory, you can recall them and run them afterward as often as desired.

2.2   Spindle Speed S, Feed Rate F and Miscellaneous Functions M

These are the soft keys in the MANUAL OPERATION and ELECTRONIC HANDWHEEL modes:

M	S	TOUCH PROBE	DATUM SET		3D ROT		TOOL TABLE

With these functions and with the override knobs on the TNC keyboard you can change and enter:

- spindle speed S
- feed rate F (only via override knob)
- miscellaneous functions M

These functions are entered directly in a part program in the PROGRAMMING AND EDITING mode.
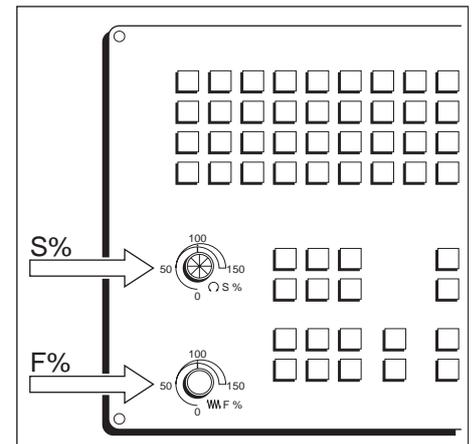
Fig. 2.4:   Knobs for spindle speed and feed rate overrides

To enter the spindle speed S:

The machine tool builder determines which spindle speeds are allowed on your TNC. Your machine manual provides more information on the available spindle speeds.

S	Select S for spindle speed.

SPINDLE SPEED S =

e.g. **1** **0** **0** **0** **ENT** **I**	Enter the desired spindle speed (for example, 1000 rpm). Press the machine START button to confirm the entered spindle speed.

The spindle speed S with the entered rpm is started with a miscellaneous function M.

To change the spindle speed S:
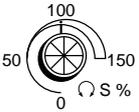
100 50 150 0 ∩ S %	Turn the knob for spindle speed override: You can vary the spindle speed from 0% to 150% of the last entered value.

The knob for spindle speed override is effective only on machines with a stepless spindle drive.

2.2   Spindle Speed S, Feed Rate F and Miscellaneous Functions M

To change the feed rate F:

In the MANUAL OPERATION mode the feed rate is set by a machine parameter.
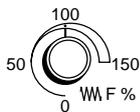
100 50 ◯ 150 0 〰 F %	Turn the knob for feed rate override. You can vary the feed rate from 0% to 150% of the set value.

To enter a miscellaneous function M:

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

M	Select M for miscellaneous function.

MISCELLANEOUS FUNCTION M =	
e.g. **6** **ENT** **I**	Enter the miscellaneous function (for example, M6). Press the START button to activate the miscellaneous function.

See Chapter 11 for a list of the miscellaneous functions.

2.3 Setting the Datum Without a 3D Touch Probe

You fix a datum by setting the TNC position display to the coordinates of a
known point on the workpiece. The fastest, easiest and most accurate
way of setting the datum is by using a 3D touch probe from HEIDENHAIN
(see page 2-14).

To prepare the TNC:

Clamp and align the workpiece.	

Insert the zero tool with known radius into the spindle.	

🖐 or ⊛	Select the MANUAL OPERATION or ELECTRONIC HANDWHEEL mode.

Ensure that the TNC is showing the actual values (see page 10-8).	
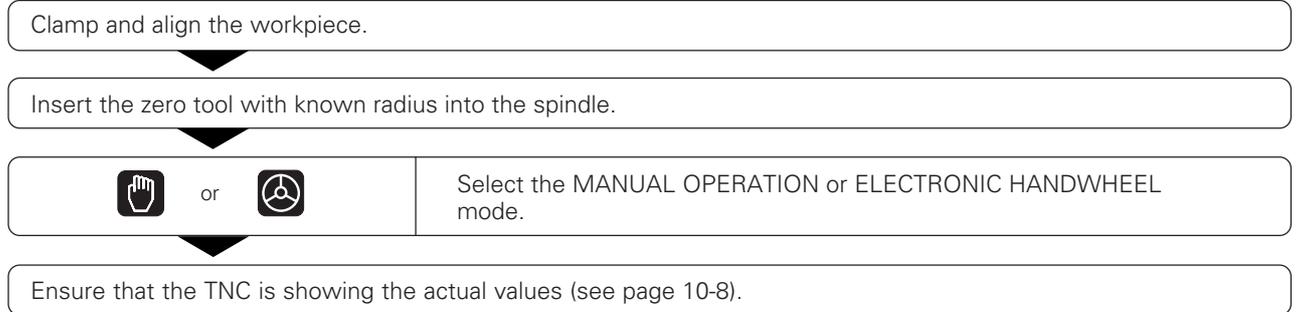
Setting the datum in the tool axis

☞ | Fragile workpiece?
If the workpiece surface must not be scratched,
you can lay a metal shim of known thickness *d*
on it. Then enter a tool axis datum value that is
larger than desired datum by the value *d*.
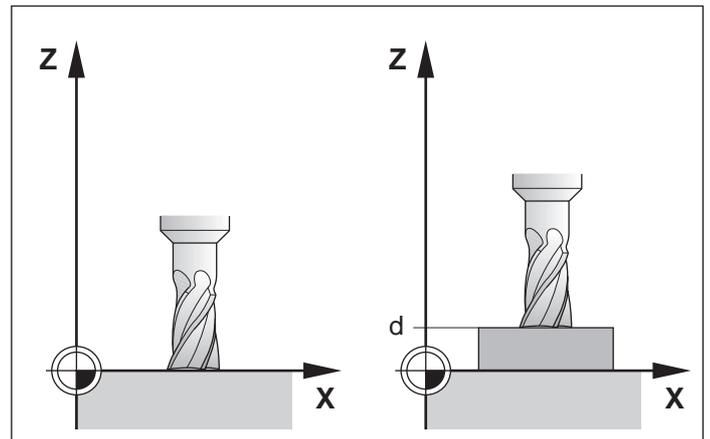
Fig. 2.5:   Workpiece setting in the tool axis; *right*, with protective shim

Move the tool until it touches the workpiece surface.	

e.g. **Z**	Select the tool axis.

ELECTRONIC HANDWHEEL only: **DATUM SET**	Select datum setting.

e.g. **0** **ENT**	Zero tool. Set the display to Z = 0 or enter the thickness *d* of the shim.
e.g. **5 0** **ENT**	Preset tool: Set the display to the length *L* of the tool, (here Z = 50 mm or enter the sum Z = L + d
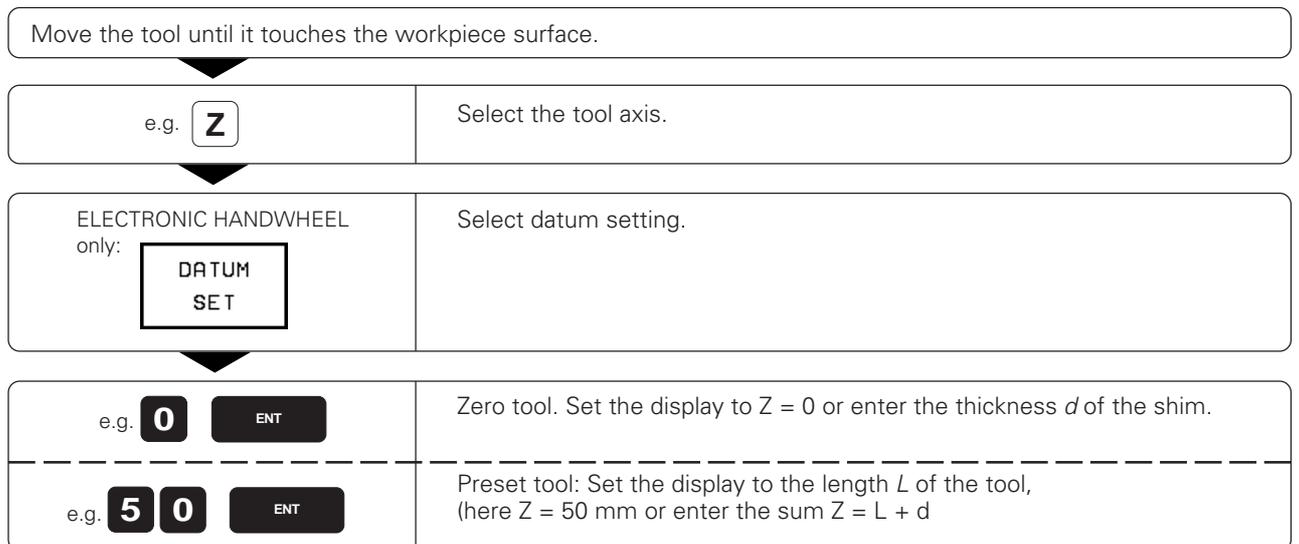
To set the datum in the working plane:
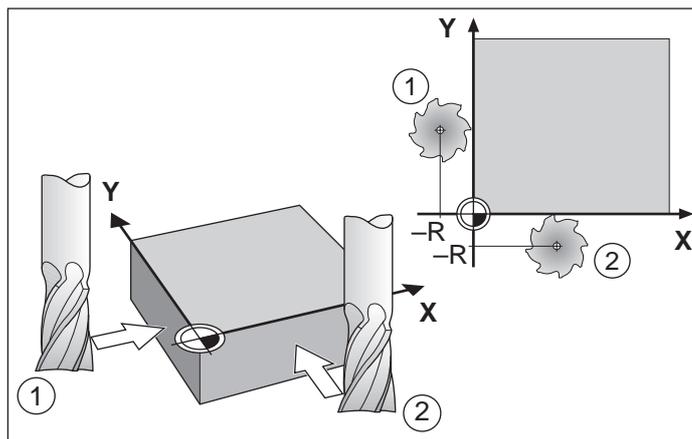
Fig. 2.6:   Setting the datum in the working plane; plan view *(upper right)*

Move the zero tool until it touches the side of the workpiece.	
e.g. **X**	Select the axis.
ELECTRONIC HANDWHEEL only: **DATUM SET**	Select datum setting.
e.g. **-/+** **5** **ENT**	Enter the position of the tool center (here, X = 5 mm) including the sign.

Repeat the process for all axes in the working plane.

The exact dialog for datum setting depends on machine parameters MP 7295 and MP 7296 (see page 11-11).

2.4 3D Touch Probes

3D Touch probe applications

Your TNC supports a HEIDENHAIN 3D touch probe.
Typical applications for touch probes:

- Compensating misaligned workpieces (basic rotation)
- Datum setting
- Measuring:
  - lengths and workpiece positions
  - angles
  - radii
  - circle centers
- Measurements during program run
- Digitizing 3D surfaces

Fig. 2.7:   3D touch probe model TS 120

- The TNC must be specially prepared by the machine manufacturerer for the use of a 3D touch probe.
- If you wish to make measurements during program run, ensure that the tool data (length, radius, axis) are taken either from the calibrated data or from the last TOOL CALL block (selection through MP 7411, see page 11-13).

After you press the machine START button, the touch probe begins executing the selected probing function. The machine tool builder sets the feed rate F at which the probe approaches the workpiece (MP6120). When the touch probe contacts the workpiece, it

- transmits a signal to the TNC (the coordinates of the probed position are stored),
- stops moving, and
- returns to its starting position at rapid traverse.

If the stylus is not deflected within the distance defined in MP 6130, the TNC displays an error message.
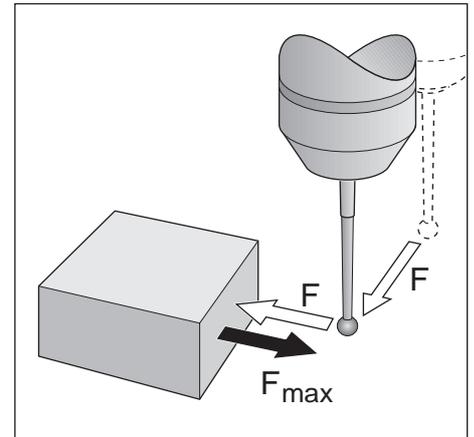
Fig. 2.8:   Feed rates during probing

To select the touch probe functions:

MANUAL OPERATION

or

ELECTRONIC HANDWHEEL

TOUCH PROBE   Select the touch probe functions.

CAL L	CAL R	PROBING ROT	PROBING POS	PROBING P	PROBING CC	END

Calibrating the 3D touch probe

The touch probe must be calibrated in the following cases:

- for commissioning
- after stylus breakage
- when the stylus is changed
- when the probing feed rate is changed
- in the case of irregularities, such as those resulting from warming of the machine.

During calibration, the TNC finds the "effective" length of the stylus and the "effective" radius of the ball tip. To calibrate the touch probe, clamp a ring gauge of known height and known inside radius to the machine table.
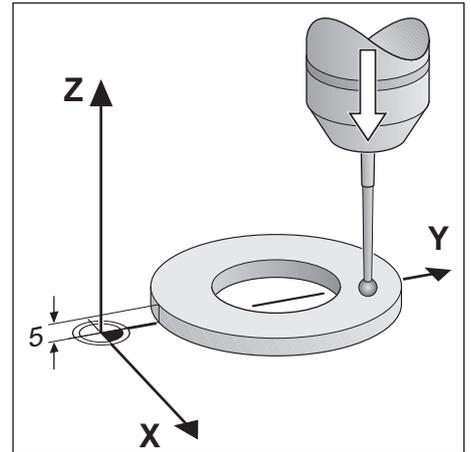
Fig. 2.9:   Calibrating the touch probe length

To calibrate the effective length:

Set the datum in the tool axis such that for the machine tool table, Z=0.
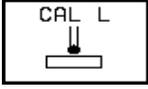
CAL L	Select the calibration function for the touch probe length.

MANUAL OPERATION

Z+ Z–

TOOL AXIS =   Z

e.g. **Z**	If necessary, enter the tool axis.
↓	Move the highlight to DATUM.
e.g. **5**	Enter the height of the ring gauge (here, 5 mm).

Move the touch probe to a position just above the ring gauge.

← or →	If necessary, change the displayed traverse direction.

I	The touch probe contacts the upper surface of the ring gauge.

To calibrate the effective radius

Position the ball tip in the bore hole of the ring gauge.

Compensating center misalignment

After the touch probe is inserted it normally needs to be exactly aligned with the spindle axis. The misalignment is measured with this calibration function and automatically compensated electronically.

For this operation the 3D touch probe is rotated by 180°. The rotation is initiated by a miscellaneous function that is set by the machine tool builder in the machine parameter MP 6160.

The center misalignment is measured after the effective ball tip radius is calibrated.
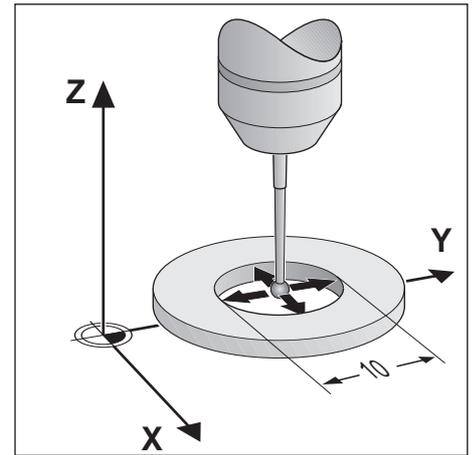
Fig. 2.10:   Calibrating the touch probe radius and determining center misalignment

CAL R ⊙	Select the calibrating function for the ball-tip radius and the touch probe center misalignment.

MANUAL OPERATION

X +   X –   Y +   Y –

TOOL AXIS =   Z

RADIUS RING GAUGE = 0

4x **I**	The touch probe contacts a position on the bore in each axis direction; the effective ball-tip radius is stored.

180°⤸							END

180°⤸	Determine the ball-tip center misalignment (or terminate the calibration function with END): rotate the touch probe by 180°.

4x **I**	The touch probe contacts one position on the bore hole for each axis direction; the touch probe center misalignment is stored.

2.4   3D Touch Probes

Displaying calibration values

The effective length and radius, and the center misalignment of the 3D touch probe are stored in the TNC for use when the touch probe is needed again. You can display the values on the screen with the soft keys CAL L and CAL R.

```
MANUAL OPERATION              PROGRAMMING
X+   X-   Y+   Y-              AND EDITING



TOOL AXIS =Z
RADIUS RING GAUGE =25
EFFECT.PROBE RADIUS =3,9996
EFFECTIVE LENGTH =+12,7836
STYLUS TIP CENTER OFSET X=+0,0051
STYLUS TIP CENTER OFSET Y=+0,0009


ACTL.    X    +25,3684   Y   -250,3600
         Z    -25,0000   B   +331,0000
         C    +12,5000
T                        F 0        M 5/9
```

```
                                    END
```

Fig. 2.11:   Menu for touch probe radius and center misalignment

Compensating workpiece misalignment

The TNC electronically compensates workpiece misalignment by computing a "basic rotation". You set the rotation angle to the desired angle with respect to the reference axis in the working plane (see page 1-13).
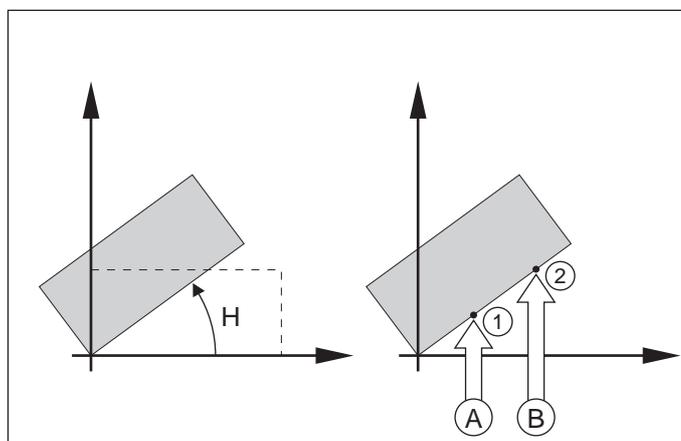
Fig. 2.12:   Basic rotation of a workpiece; probing procedure for compensation *(right)*. The broken line is the nominal position, the angle *H* is being compensated.

PROBING ROT	Press the PROBING ROT soft key.

ROTATION ANGLE =	
e.g. **0** **ENT**	Enter the nominal value of the rotation angle.

Move the ball tip *(A)* to a starting position near the first touch point *(1)*.

.
.
.
.
.

X +	X −	Y +	Y −

← or →	Select the probe direction.

I	Probe the workpiece.

Move the ball tip *(B)* to a starting position near the second touch point *(2)*.

I	Probe the workpiece.
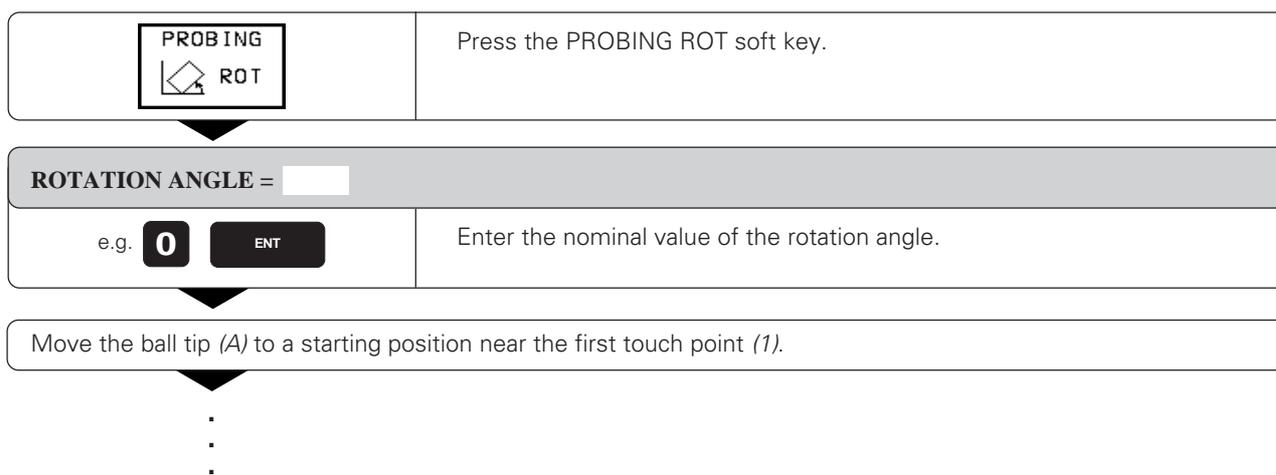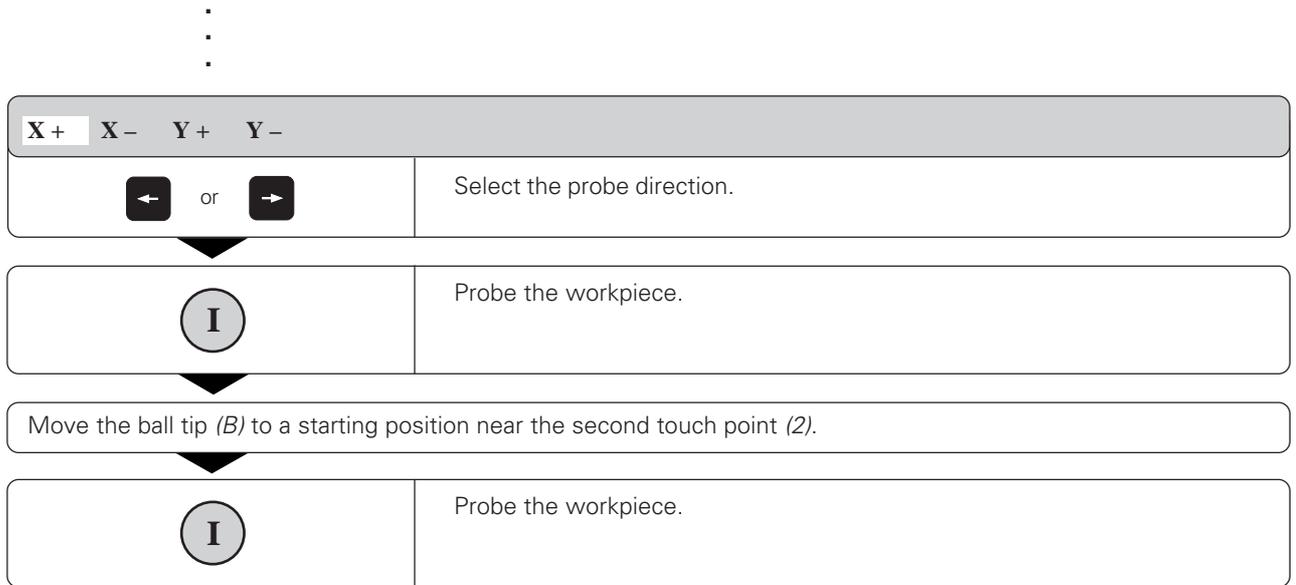
A basic rotation is kept in non-volatile storage and is effective for all subsequent program runs and graphic simulation.

Displaying basic rotation

The angle of the basic rotation appears after ROTATION ANGLE whenever PROBING ROT is selected. It is also shown in the additional status display (see page 1-27) under ROTATION.

In the status display, a symbol is shown for a basic rotation whenever the TNC is moving the axes according to a basic rotation.

```
MANUAL  OPERATION              PROGRAMMING
X+   X−   Y+   Y−              AND EDITING




ROTATION  ANGLE  =+12,5

ACTL.     X   +25,3684     Y   −250,3600
          Z   −25,0000     B   +331,0000
          C   +12,5000
T                          F 0        M 5/9
                                          END
```

Fig. 2.13:   Displaying the angle of an active basic rotation.

To cancel a basic rotation:

PROBING ROT	Select the probing function with the soft key PROBING ROT.

ROTATION ANGLE =	
0 ENT	Set the rotation angle to 0.

END	Terminate the probing function.
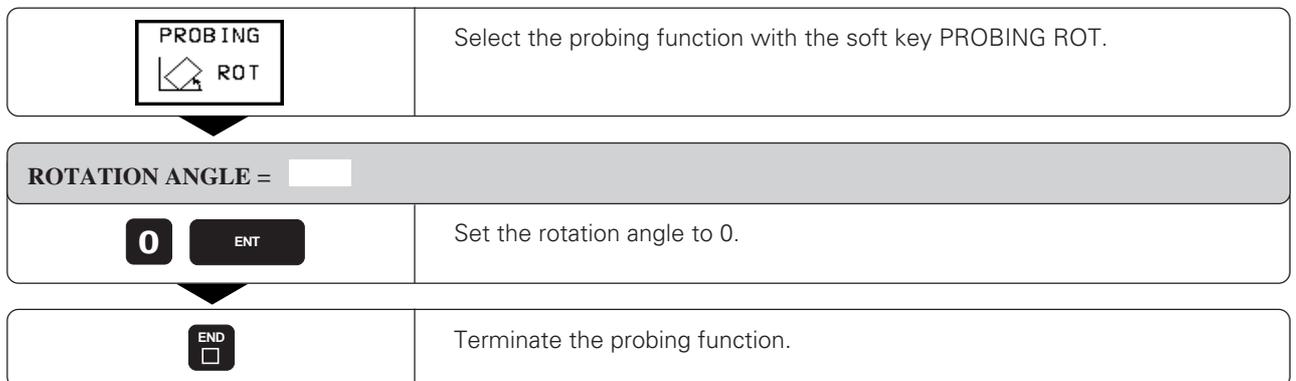
2.5 Setting the Datum with a 3D Touch Probe

The following functions are available for setting the datum on an aligned workpiece:

- Datum setting in any axis with PROBING POS
- Defining a corner as datum with PROBING P
- Setting the datum at a circle center with PROBING CC

To set the datum in an axis:
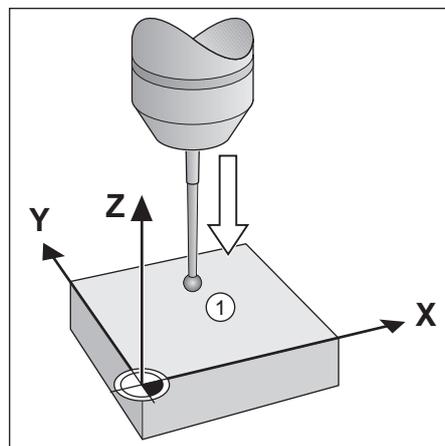
Fig. 2.14:   Probing for the datum in the Z axis

PROBING POS	Select the probing function with the soft key PROBING POS.

Move the touch probe to a position near the touch point.

X + X – Y + Y – Z + **Z –**	
← or →	Select the probe axis and direction in which you wish to set the datum, such as Z in direction Z–.

I	Probe the workpiece.

e.g. **0** **ENT**	Enter the nominal coordinate of the datum.
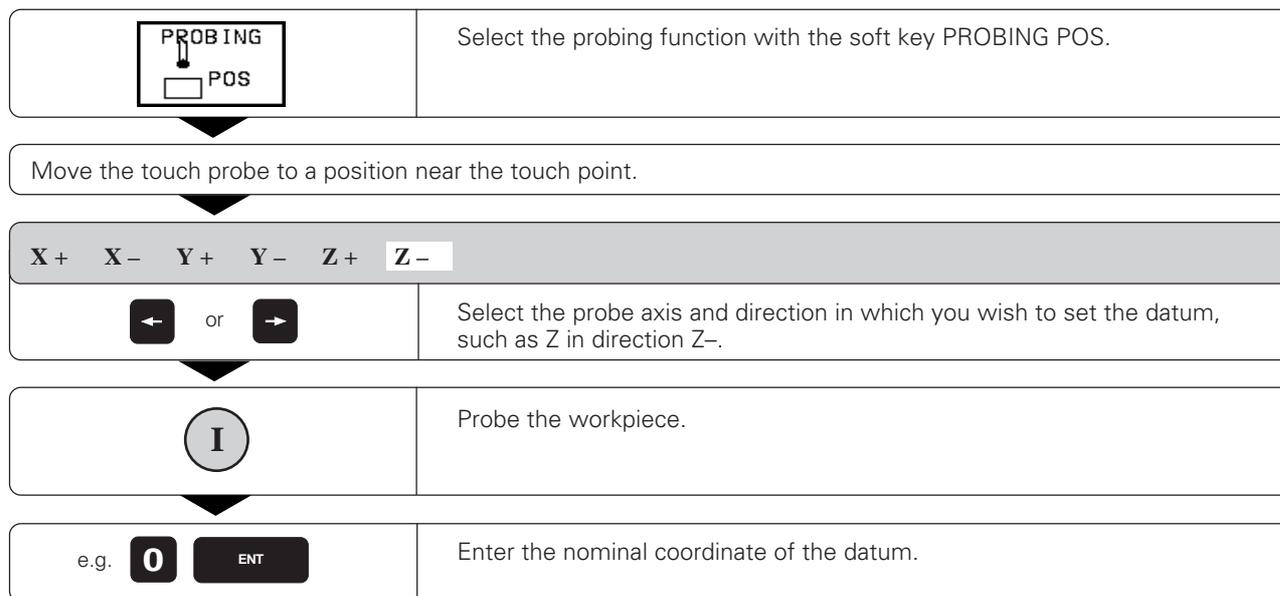
Corner as datum
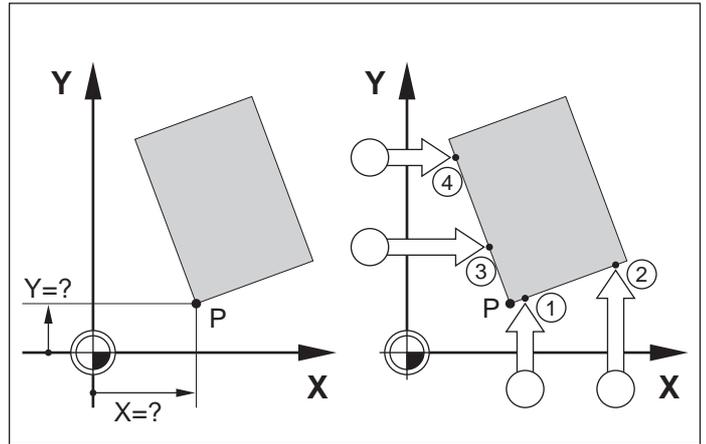
Fig. 2.15:   Probing procedure for finding coordinates of corner P

PROBING P⌐_	Select the probing function with the soft key PROBING P.

To use the points that were already probed for a basic rotation:

TOUCH POINTS OF BASIC ROTATION?	
ENT	Transfer the touch point coordinates to memory.

Move the touch probe to a starting position near the first touch point of the side that was not probed for basic rotation.

X + X – Y + Y –	
← or →	Select the probe direction.

I	Probe the workpiece.

Move the touch probe to a position near the second touch point on the same side.

I	Probe the workpiece.

DATUM	
e.g. 0 ENT	Enter the first coordinate of the datum point (for example, in the X axis).

2.5   Setting the Datum with a 3D Touch Probe

:
:
:

↓	Select the second coordinate.

DATUM

e.g. **0** ENT	Enter the second coordinate of the datum (for example, in the Y axis).

END	Terminate the probing function.
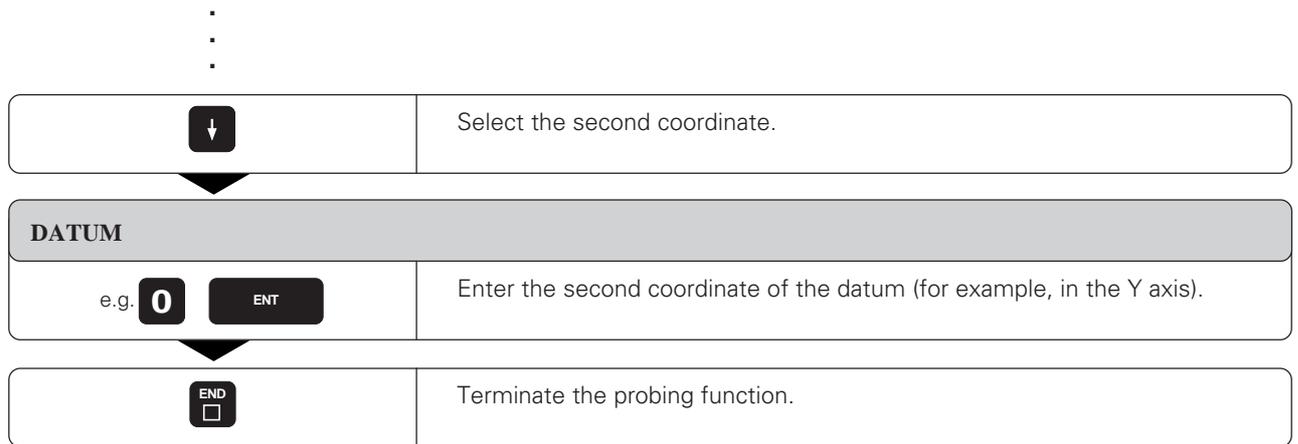
If you do *not* wish to use the points that were already probed for a basic rotation:

TOUCH POINTS OF BASIC ROTATION?

NO ENT	Ignore the previous touch point coordinates.

Probe both workpiece sides twice each.

Enter the coordinates of the datum.
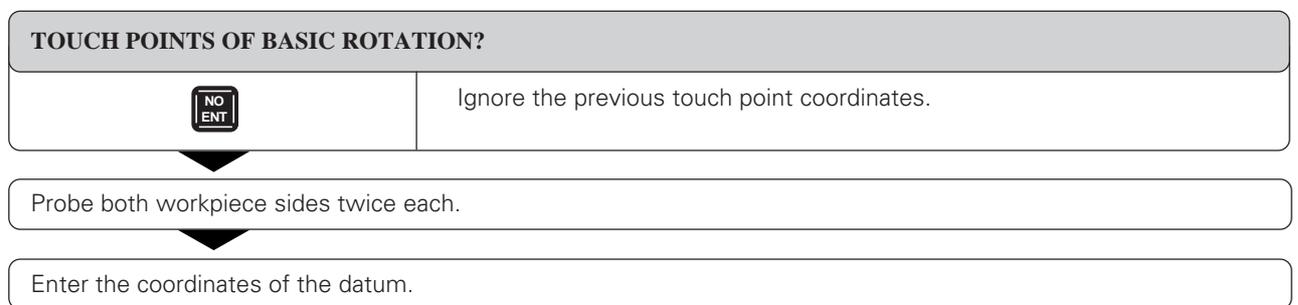
Circle center as datum

With this function you can set the datum at the center of bore holes, circular pockets, cylinders, journals, circular islands, etc.
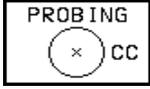
PROBING ⊗ CC	Select the probing function with the soft key PROBING CC.

Inside circle

The TNC automatically probes the inside wall in all four coordinate axis directions.

For incomplete circles (circular arcs) you can choose the appropriate probing directions.
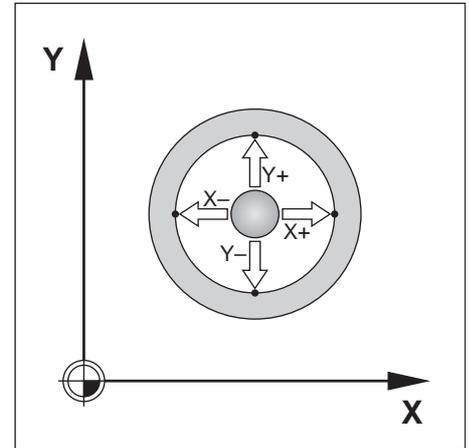
Fig. 2.16: Probing the inside of a cylindrical surface to find the center

Move the touch probe to a position approximately in the center of the circle.	

X + **X –** **Y +** **Y –**	
4 x **I**	The probe touches four points on the inside of the circle.
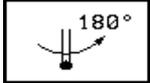
180° 4 x **I**	Rotate the touch probe by 180° if you are probing to find the stylus center (only available on machines with spindle orientation, depending on MP6160). Then probe another four points on the inside of the circle.
END	Terminate the probing function for finding the stylus center.

2.5   Setting the Datum with a 3D Touch Probe

. . .

DATUM	
e.g. **0** **ENT**	Enter the first coordinate of the datum (for example, in the X axis).

↓	Select the second coordinate.

DATUM	
e.g. **1** **0** **ENT**	Enter the second coordinate of the datum (for example, in the Y axis).

END	Terminate the probing function.

Outside circle

Fig. 2.17: Probing the outside of a cylindrical surface to find the center

Move the touch probe to the starting position near the first touch point *(1)* outside of the circle.	

X + X – Y + Y –	
← or **→**	Select the probing direction.

I	Probe the workpiece.

Repeat the probing process for points 2, 3 and 4 (see illustration).	

Enter the coordinates of the datum.	
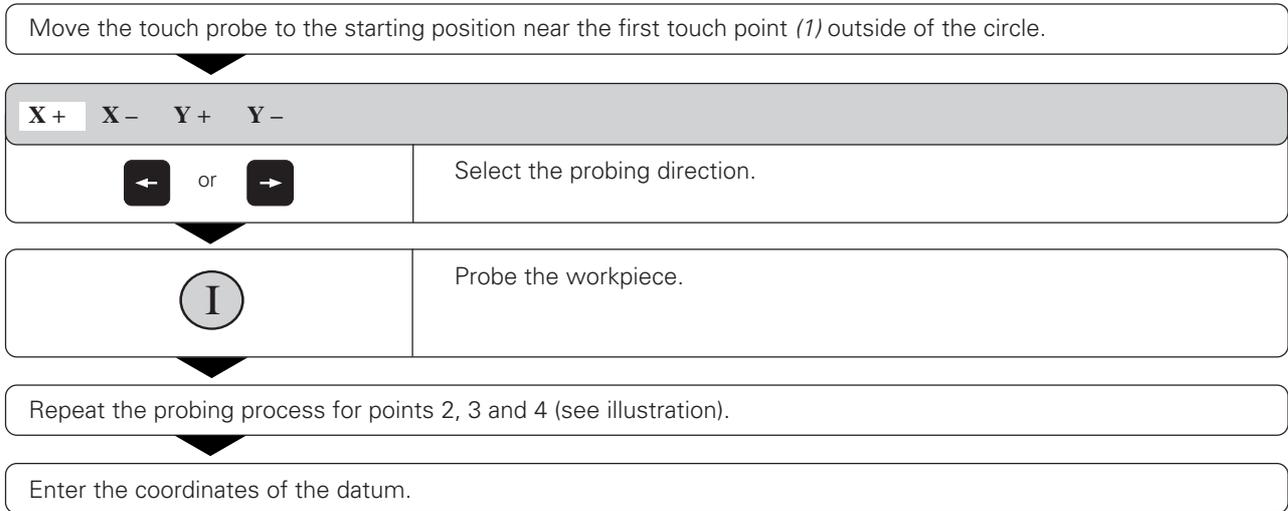
After the probing procedure is completed, the TNC displays the coordinates of the circle center and the circle radius PR.

2.5   Setting the Datum with a 3D Touch Probe

Setting datum points over holes

A second soft-key row provides soft keys for using holes to set datums.

The touch probe is used in the same way as in the "circle center as datum" function (see page 2-17). First pre-position it in the approximate center of a hole, then press the machine START button to automatically probe four points in the hole.

Move the touch probe to the next hole and have the TNC repeat the probing procedure until all the holes have been probed to set datums.

```
MANUAL OPERATION                    PROGRAMMING
                                    AND EDITING




ACTL.    X    +25,3684      Y   -250,3600
         Z    -25,0000      B   +331,0000
         C    +12,5000
T                          F 0          M 5/9
          PROBING          PROBING  PROBING
            ROT               P        CC      END
```
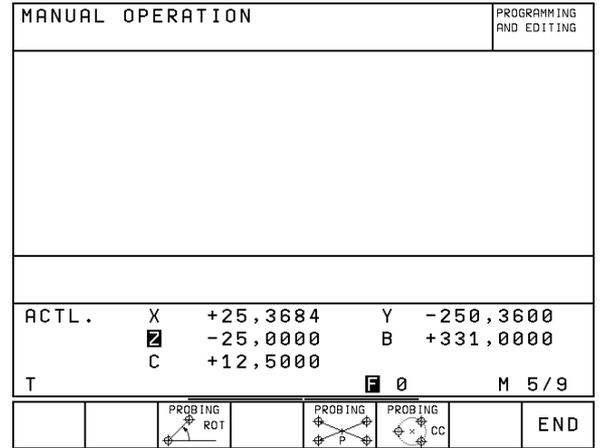
Fig. 2.18:   Second soft-key row for TOUCH PROBE

Function	Soft key
Basic rotation from 2 holes: The TNC measures the angle between the line connecting the centers of two holes and a nominal angular position (angle reference axis).	PROBING ROT
Datum from 4 holes: The TNC calculates the intersection of the line connecting the first two probed holes with the line connecting the last two probed holes. If a basic rotation was already made from the first two holes, these holes do not need to be probed again.	PROBING P
Circle center from 3 holes: The TNC calculates a circle that intersects the centers of all three holes, and finds the center.	PROBING CC

2.6 Measuring with a 3D Touch Probe

With a 3D touch probe you can determine

- position coordinates, and from them,
- dimensions and angles on the workpiece.

To find the coordinates of a position on an aligned workpiece:

PROBING POS	Select the probing function with the soft key PROBING POS.

Move the touch probe to a position near the touch point.

X +	X –	Y +	Y –	Z +	Z –

← or →	Select the probe direction and axis of the coordinate.

I	Probe the workpiece.
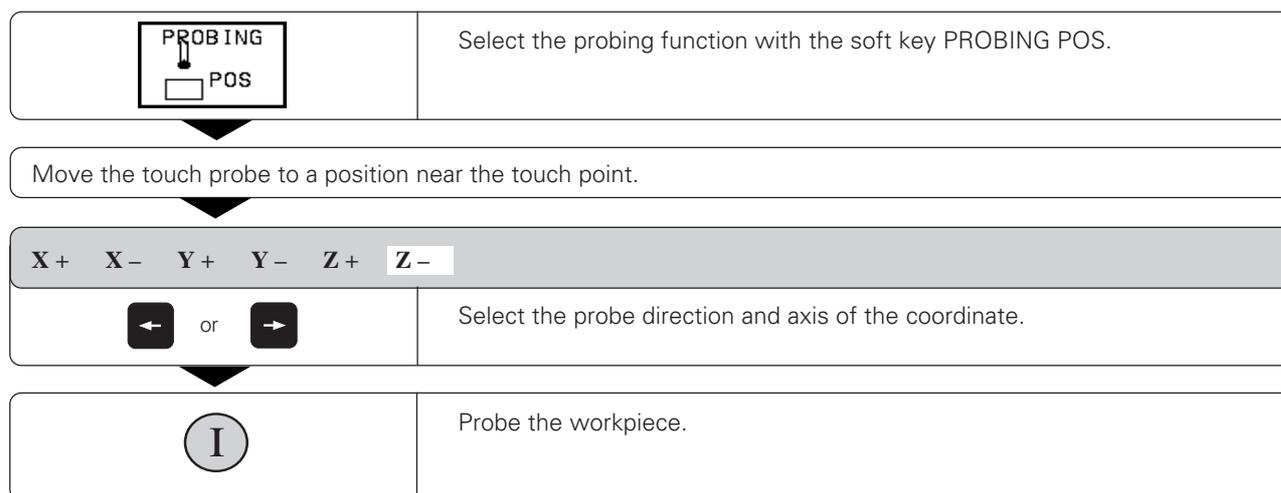
The TNC shows the coordinates of the touch point as DATUM.

Finding the coordinates of a corner in the working plane

Find the coordinates of the corner point as described under "Corner as datum." The TNC displays the coordinates of the probed corner as DATUM.

Measuring workpiece dimensions
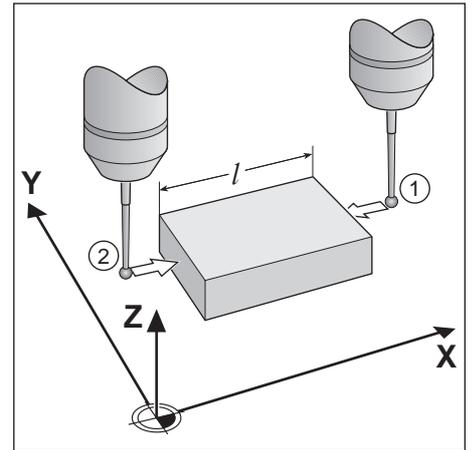
Fig. 2.19:   Measuring lengths with the 3D touch probe

PROBING ▯ POS	Select the probing function with the soft key PROBING POS.

Move the touch probe to a position near the first touch point *(1)*.

X +   X –   Y +   Y –   Z +   Z –

← or →	Select the probing direction with the cursor keys.

I	Probe the workpiece.

If you will need the current datum later, write down the value that appears in the DATUM display.

DATUM

0 ENT	Set the DATUM to 0.

END ▯	Terminate the dialog.

PROBING ▯ POS	Select the probe function again with the soft key PROBING POS.

Move the touch probe to a position near the second touch point *(2)*.

.
.
.

.
.
.

X + X – Y + **Y –** Z + Z –	
← or →	Select the probe direction with the cursor keys – same axis as for *1*.

I	Probe the workpiece.

The value displayed as DATUM is the distance between the two points.

To return to the datum that was active before the length measurement:

PROBING POS	Select the probing function with the soft key PROBING POS.

Probe the first touch point again.

Set the DATUM to the value that you wrote down previously.

END	Terminate the dialog.
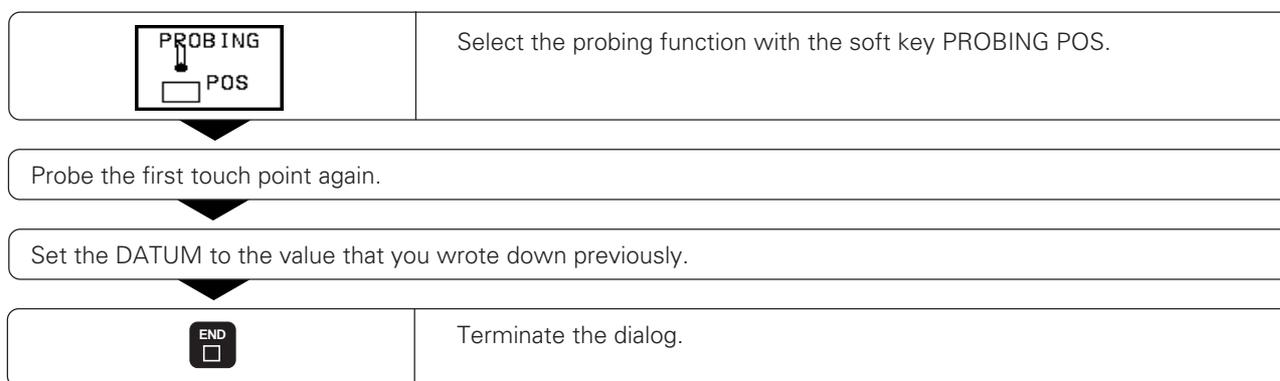
Measuring angles

You can also use the touch probe to measure angles in the working plane.
You can measure

• the angle between the angle reference axis and a workpiece side, or
• the angle between two sides.

The measured angle is displayed as a value of maximum 90°.

To find the angle between the angle reference axis and a side of the workpiece:

PROBING ROT	Select the probing function with the soft key PROBING ROT.

ROTATION ANGLE
If you will need the current basic rotation later, write down the value that appears under ROTATION ANGLE.

Make a basic rotation with the side of the workpiece (see section "Compensating workpiece misalignment").

.
.
.

.
.
.
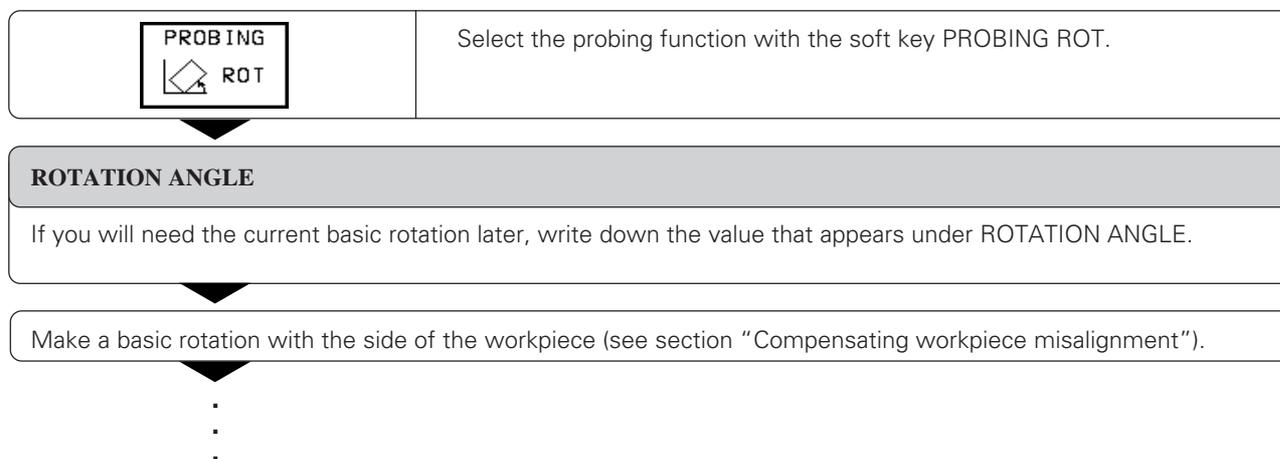
| **PROBING** ROT | Display the angle between the angle reference axis and the side of the workpiece as the ROTATION ANGLE. |

Cancel the basic rotation.

- -

To restore the previous basic rotation:
Set the ROTATION ANGLE to the value you wrote down previously.

To measure the angle between two sides of a workpiece:
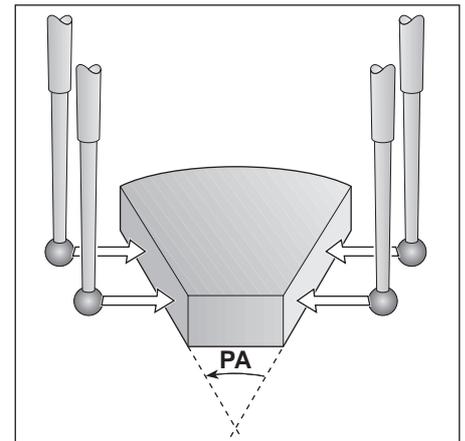
Fig. 2.20:   Measuring the angle between two sides of a workpiece

| **PROBING** ROT | Select the probing function with the PROBING ROT soft key. |

ROTATION ANGLE

If you will need the current basic rotation later, write down the value that appears under ROTATION ANGLE.

Make a basic rotation for the first side (see "Compensating workpiece misalignment").

Probe the second side as for a basic rotation, but do *not* set the ROTATION ANGLE to zero!
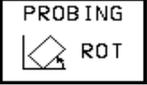
| **PROBING** ROT | The angle PA between the two sides appears under ROTATION ANGLE. |

Cancel the basic rotation.

- -

To restore the previous basic rotation:
Set the ROTATION ANGLE to the value you wrote down previously.

2.7 Tilting the Working Plane

The functions for tilting the working plane are interfaced to the TNC and the machine tool by the machine tool builder. On the TNC 426, the machine tool builder also determines whether the entered angles are interpreted as coordinates of the tilt axes or as solid angles. Your machine manual provides more detailed information on this subject.

The working plane is always tilted around the active datum.

The TNC supports machine tools with swivel heads (the tool is tilted) and/or tilting tables (the workpiece is tilted).

The program is written as usual in a main plane, such as the X/Y plane, but is executed in a plane that is tilted relative to the main plane.

Typical applications:

• Oblique holes
• Contours in an oblique plane

There are two ways to tilt the working plane:

• 3D ROT soft key in the MANUAL OPERATION and ELECTRONIC HANDWHEEL operation modes
• Cycle G80 WORKING PLANE in the part program (see page 8-55)

The tilting functions are coordinate transformations. The transformed tool axis (i.e., as calculated by the TNC) always remains parallel to the actual tool axis (the axis being positioned). The working plane is always perpendicular to the direction of the tool axis.

When tilting the working plane, the TNC differentiates between two machine types:

• Machines with tilting tables
• Machines with swivel heads

Machines with tilting tables:

• You must bring the **workpiece** into the desired position for machining by positioning the tilting table, for example with a G00 block.

• The position of the transformed tool axis **does not change** in relation to the machine-based coordinate system. Thus if you rotate the table— and therefore the workpiece—by 90° for example, the coordinate system **does not rotate**. If you press the Z+ axis direction button in the MANUAL OPERATION mode, the tool moves in Z+ direction.

• In calculating the transformed coordinate system the TNC considers only the mechanically influenced offsets of the particular tilting table (the so-called "translational" components).

Machines with swivel heads:

- You must bring the **tool** into the desired position for machining by positioning the swivel head, for example with a G00 block.

- The position of the transformed tool axis (like the position of the tool) **changes** in relation to the machine-based coordinate system. Thus if you rotate the swivel head—and therefore the tool—in the B axis by +90° for example, the coordinate system **rotates also**. If you press the Z+ axis direction button in the MANUAL OPERATION mode, the tool moves in X+ direction of the machine-based coordinate system.

- In calculating the transformed coordinate system the TNC considers the mechanically influenced offsets of the particular swivel head (the so-called "translational" components) **and** the offsets caused by tilting of the tool (3D tool length compensation).

Traversing reference points with tilted axes

When axes are tilted, the reference points are traversed by pressing the machine axis direction buttons. The TNC interpolates the tilted axes. Make sure that the tilting function is active in the manual operating mode and that the actual angle value of the tilted axis was correctly entered in the menu (see page 2-26).

Setting the datum in a tilted coordinate system

After you have positioned the tilted axes, set the datum in the same way as for non-tilted axes: either manually by touching the workpiece with the tool (see page 2-7), or (much more easily) by allowing the part program to automatically set the datum with the aid of the HEIDENHAIN 3D touch probe (see page 2-14).

The TNC then converts the datum for the tilted coordinate system. The angular values for this calculation are taken from the menu for manual tilting, regardless of whether the tilting function is active or not.

> The angular values entered in the menu for manual tilting (see page 2-26) must correspond to the actual position(s) of the tilted axis or axes. The TNC will otherwise calculate a wrong datum.

Position display in the tilted system

The positions displayed in the status window (NOML and ACTL) are in the tilted coordinate system.

Limitations on working with the tilting function

- The touch probe function BASIC ROTATION cannot be used.

- PLC positioning (determined by the machine tool builder) is not possible.

2.7   Tilting the Working Plane

To activate manual tilting:

3D ROT	Select menu for manual tilting.
↑ or ↓	Select the tilt axis.
e.g. 4 5 ENT	Enter the tilt angle (for example, 45°).
ENT	Set TILT WORKING PLANE to ACTIVE.
END	Conclude input.
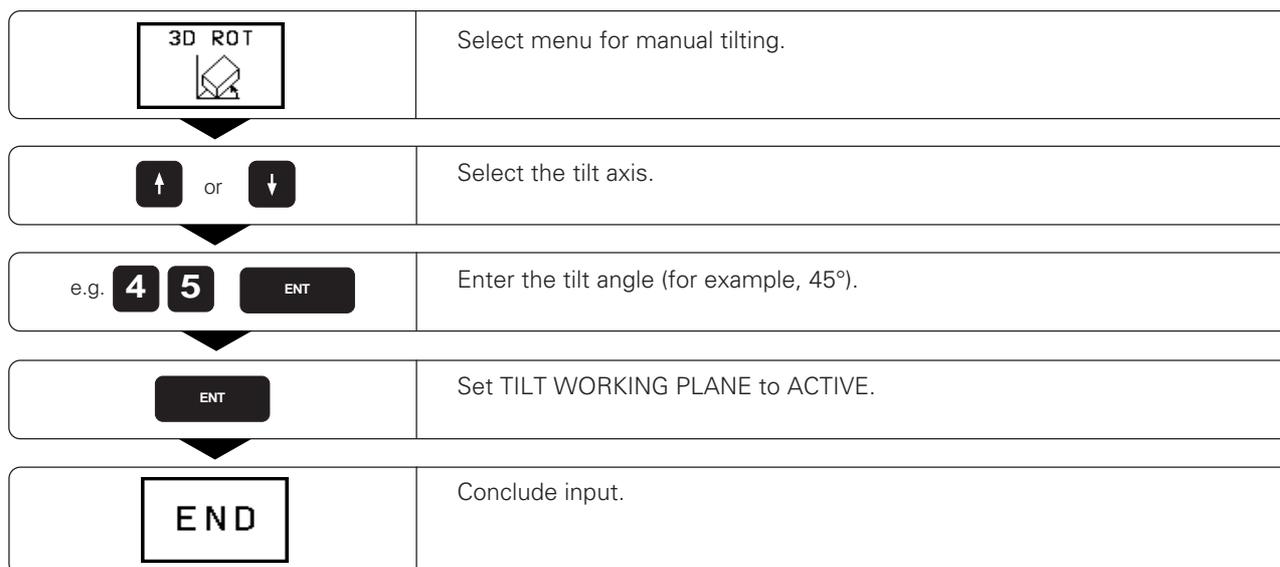
A symbol for the tilted plane is shown in the status display
whenever the TNC is moving the machines axes in the
tilted plane.

> If you have set the function TILT WORKING PLANE to ACTIVE in the PROGRAM RUN mode of operation, the tilt
> angle entered in the menu becomes effective in the first block of the part program to be executed. If you have
> entered cycle G80 WORKING PLANE in the part program, the angular values defined in the cycle become effective
> immediately after cycle definition. Angular values entered in the menu will be overwritten.

To reset:

Set TILT WORKING PLANE to INACTIVE.

```
MANUAL OPERATION              PROGRAMMING
                              AND EDITING

TILT WORKING PLANE
PROGRAM RUN:          INACTIVE
MANUAL OPERATION     ACTIVE


B = +12,5        °
C = +90          °



ACTL.     X   +65,6792    Y   -21,5938
          Z   +114,4964   B   +12,5000
          C   +90,0000
T         ⬠            F 0       M 5/9

                                      END
```

Fig. 2.21:   Menu for manual tilting in the MANUAL
OPERATION mode

3.1 Test Run

In the TEST RUN mode of operation the TNC checks programs and program sections for the following errors (without moving the machine axes):

- Geometrical incompatibilities
- Missing information
- Impossible jumps
- Violation of the machine's working space

The following functions can be used in the TEST RUN operating mode:

- Blockwise test run
- Interrupt test at any block
- Block skip
- Blockwise transfer of very long programs from external storage media
- Graphic simulation
- Measurement of machining time
- Additional status display

To run a program test:

- If the central tool file is active, the tool table with which the program test is to be run must have status S (see page 1-31).
- With the MOD function DATUM SET, you can activate work space monitoring for the test run (see page 10-6).

TEST RUN

Select the program in the file directory.

| GOTO | 0 | ENT | Go to the beginning of the program. |

Function	Soft key
Test the entire program	START
Test each program block individually	START SINGLE □
Show the blank form and test the entire program	RESET + START
Interrupt the test run	STOP

To run a program test up to a certain block:

With the STOP AT N function the TNC does a test run up to the block
with block number N.

Select the TEST RUN mode and go to the program beginning.	

STOP AT N	Select a partial test run.

STOP AT: N = **PROGRAM** = **REPETITIONS** =	N70 D00 D18 P01 +90 * N80 D00 D9 P01 +0 * STOP AT: N = 120 PROGRAM = 3815.I REPETITIONS = 1 63° START SINGLE OFF/ON

e.g. **5** **ENT**	Enter the block number N at which you want the test to stop.
e.g. **1 2 3** **ENT**	Enter the name of the program that contains block number N.
e.g. **1** **ENT**	If N is located in a program section repeat, enter the number of repeats that you want to run.

START	Test the program up to the entered block.
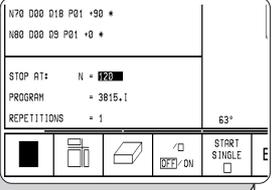
The display functions for test run

In the TEST RUN operating mode the TNC offers functions for displaying a
program in pages.

◁ or ▷	Shift the soft-key row.

PAGE ⇧	PAGE ⇩	BEGIN TEXT	END TEXT				/□ OFF/ON

Function	Soft key
Go back in the program by one screen	PAGE ⇩
Go forward in the program by one screen	PAGE ⇧
Go to the beginning of the program	BEGIN TEXT
Go to the end of the program	END TEXT

3.2 Program Run

In the PROGRAM RUN / FULL SEQUENCE mode of operation the TNC executes a part program continuously to its end or up to a program stop.

In the PROGRAM RUN / SINGLE BLOCK mode of operation you must start each block separately by pressing the machine START BUTTON.

The following functions can be used during a program run:

- Interrupt program run
- Start program run from a certain block
- Blockwise transfer of very long programs from external storage
- Block skip
- Editing and using the tool table TOOL.T
- Checking/changing Q parameters
- Graphic simulation
- Additional status display

To run a part program:

- Clamp the workpiece to the machine table.
- Set the datum.
- Select the necessary tables and pallet files.

PROGRAM RUN / SINGLE BLOCK	
or	
PROGRAM RUN / FULL SEQUENCE	
Select the part program and the necessary tables and pallet files in the file directory.	

GOTO **0** **ENT**	Go to the first block of the program.
I	Run the program.
Only in mode PROGRAM RUN / SINGLE BLOCK **I** for each block	Run each block of the part program separately.
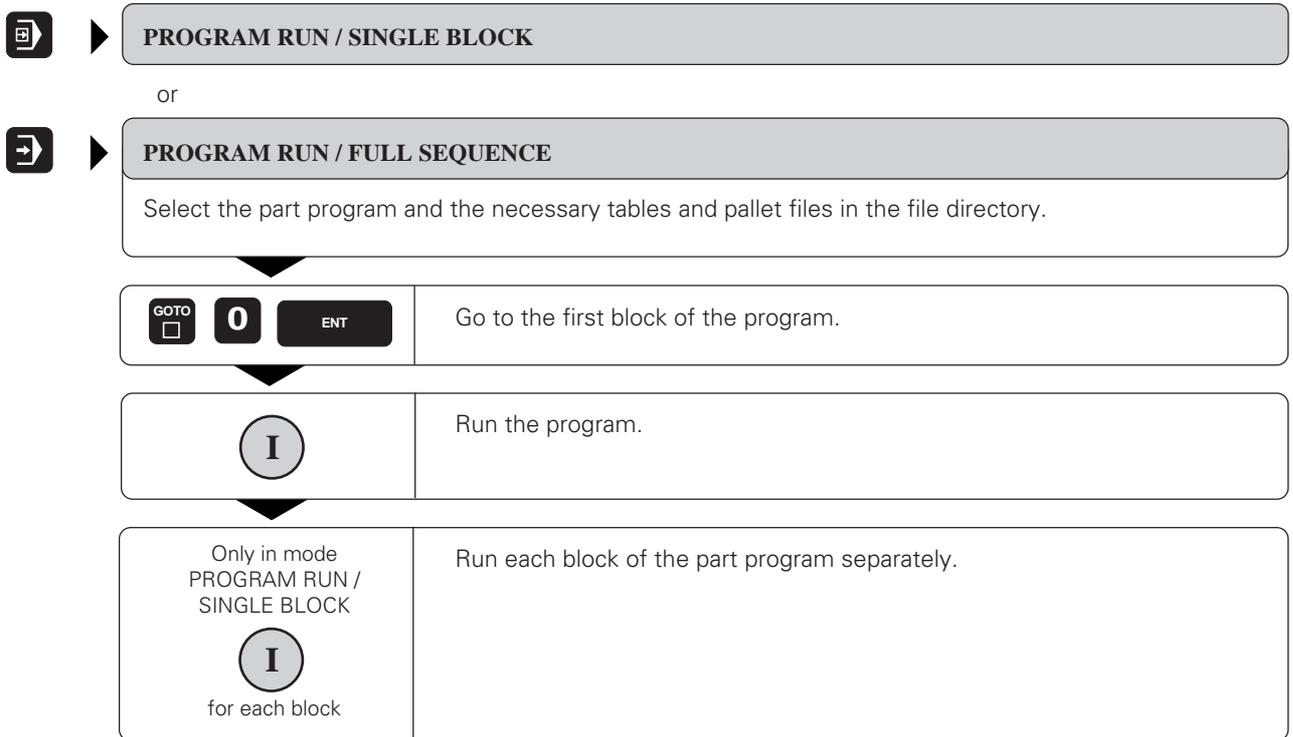
You can adjust the feed rate and spindle speed with the override knobs.

Interrupting machining

There are several ways to interrupt a program run:

• Programmed interruptions
• Machine STOP key
• Switching to PROGRAM RUN / SINGLE BLOCK

If the TNC registers an error during program run, it automatically interrupts the machining process.

Programmed interruptions

Interruptions can be programmed directly in the part program. Program run is interrupted at a block containing one of the following entries:

• G38
• Miscellaneous function M0, M02 or M30
• Miscellaneous function M06 (determined by the machine tool builder)

To interrupt or abort machining immediately:

The block which the TNC is currently executing is not completed.

◎	Interrupt machining.
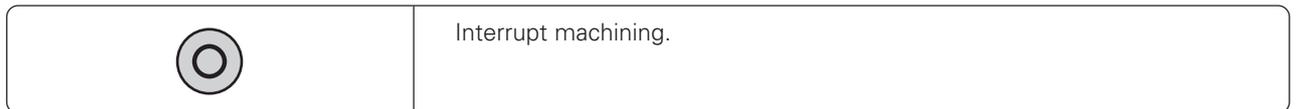
The ✳ symbol in the status display blinks.

Program run can be aborted with the INTERNAL STOP function.

INTERNAL STOP	Abort machining.
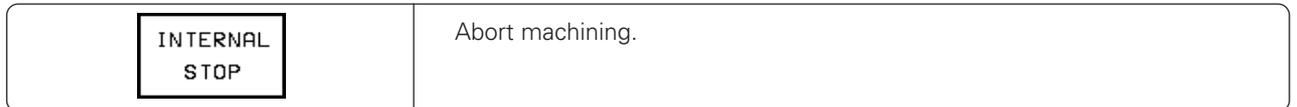
The ✳ symbol in the status display goes out.

To interrupt machining at the end of the current block:

You can interrupt the program run at the end of the current block by switching to the PROGRAM RUN / SINGLE BLOCK mode.

▣	Select PROGRAM RUN / SINGLE BLOCK.

Moving machine axes during an interruption

You can move the machine axes during a program interruption in the same way as in the MANUAL OPERATION mode. Simply enable the machine axis direction buttons by pressing the MANUAL OPERATION soft key.

> **Danger of collision**
> If you interrupt program run while the working plane is tilted, you can change from a tilted to an untilted coordinate system, and vice-versa, by pressing the 3D ON/OFF soft key. The functions of the axis direction buttons, the electronic handwheel and the positioning logic for return to contour are then evaluated by the TNC. When retracting the tool make sure the correct coordinate system is active and the angular values of the tilt axes are entered in the 3D ROT menu (see page 2-26).

Example: retracting the spindle after tool breakage

◎	Interrupt machining.
MANUAL OPERATION	Enable the machine axis direction buttons.
e.g. **Y**	Move the axes with the machine axis direction buttons.
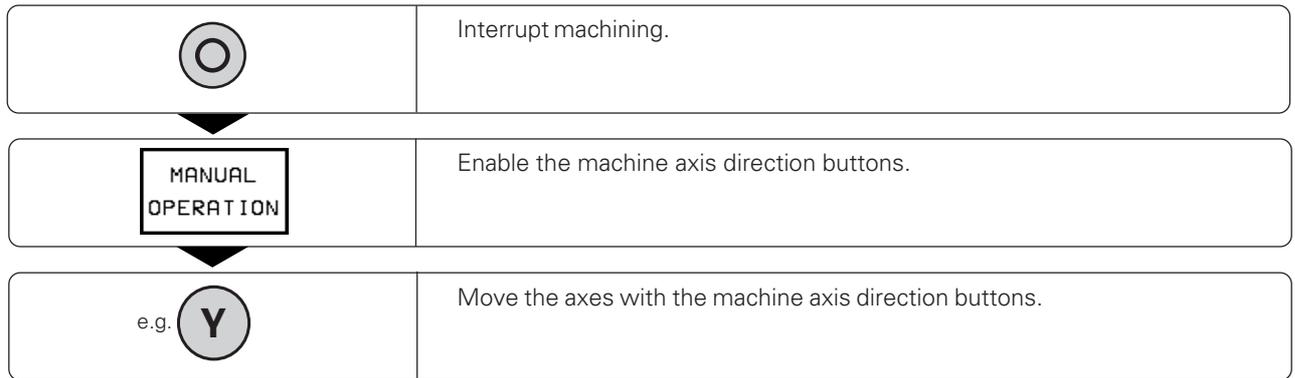
> On some machines you may have to press the machine START button after the MANUAL OPERATION soft key to enable the axis direction buttons. Refer to the operating manual of your machine tool for further information.

Resuming program run after an interruption

> • If a program run is interrupted during a fixed cycle, the program must be resumed from the beginning of the cycle. This means that some machining operations will be repeated.
> • If a program run is interrupted during execution of a subprogram or a program section repeat, use the RESTORE POS AT N function to return to the position at which the program run was interrupted.

When a program run is interrupted, the TNC stores:

• The data of the last tool called
• Active coordinate transformations
• The coordinates of the circle center that was last defined

The stored data are used for returning the tool to the contour after manual machine axis positioning during an interruption (RESTORE POSITION).

Resuming program run with the START button

You can resume program run by pressing the START button if the program was interrupted in one of the following ways:

• The machine STOP button was pressed
• A programmed interruption

3.2   Program Run

Resuming program run after an error

• If the error message is *not blinking:*

Remove the cause of the error.

CE	Clear the error message from the screen.

Restart the program, or resume program run at the place at which it was interrupted.
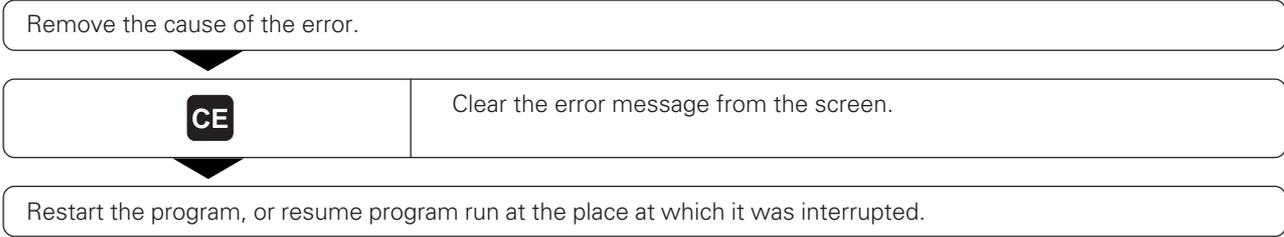
• If the error message is *blinking:*

	Switch off the TNC and the machine.

Remove the cause of the error.

Start again.
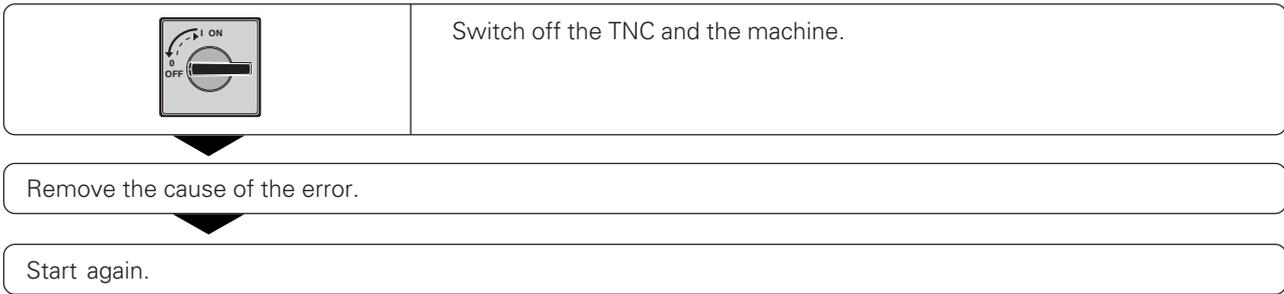
• If you cannot correct the error:

Write down the error message and contact your repair service agency.

Mid-program startup

> The RESTORE POS AT N feature must be enabled and adapted by the machine tool builder. Refer to the operating manual of your machine tool for further information.

With the RESTORE POS AT N feature (block scan) you can start a part program at any block you desire. The TNC scans the program blocks up to that point. Machining can be graphically simulated.

If a part program has been interrupted with an INTERNAL STOP, the TNC automatically offers the interrupted block N for mid-program startup.

> - Mid-program startup must not begin in a subprogram.
> - All necessary programs, tables and pallet files must be selected in a program run mode of operation.
> - If the part program contains a programmed interruption before the startup block, the block scan is interrupted. Press the machine START button to continue the block scan.
> - After a block scan, return the tool to the calculated position with RESTORE POSITION.
> - If you are working with nested programs, you can use MP7680 to define whether the block scan is to begin at block 0 of the main program, or at block 0 of the last interrupted program.
> - If the working plane is tilted, you can use the 3D ON/OFF soft key to define whether the TNC is to return to the contour in a tilted or in an untilted coordinate system.

GOTO **0** **ENT**	Go to the first block of the current program to start a block scan.
RESTORE POS. AT N	Select mid-program startup.
START-UP AT: N = **PROGRAM** = **REPETITIONS** =	*(display: N40 I1 G17 S1500 * / START-UP AT: N = 60 / PROGRAM = 3805.I / REPETITIONS = 1 / ACTL. X +132,68 / Z +160.25)*
e.g. **1 8** **ENT**	Enter the block number N at which the block scan should end.
e.g. **1 2 3 4** **ENT**	Enter the name of the program containing the block N.
e.g. **4** **ENT**	If block N is located in a program section repetition, enter the number of repetitions to be calculated in the block scan.
I	Start the block scan.
RESTORE POSITION	Return to the contour (see next page).

Returning to the contour

With the RESTORE POSITION function, the TNC returns the tool to the workpiece contour in the following situations:

- Return to contour after the machine axes were moved during a program interruption
- Return to the position that was calculated for mid-program startup

RESTORE POSITION	Select a return to contour.

↓

POSIT. LOGIC ⓘ	Move the axes in the sequence that the TNC suggests on the screen.
RESTORE Z ⓘ RESTORE X ⓘ ⋮	Move the axes in any sequence.

↓

ⓘ	Resume machining.
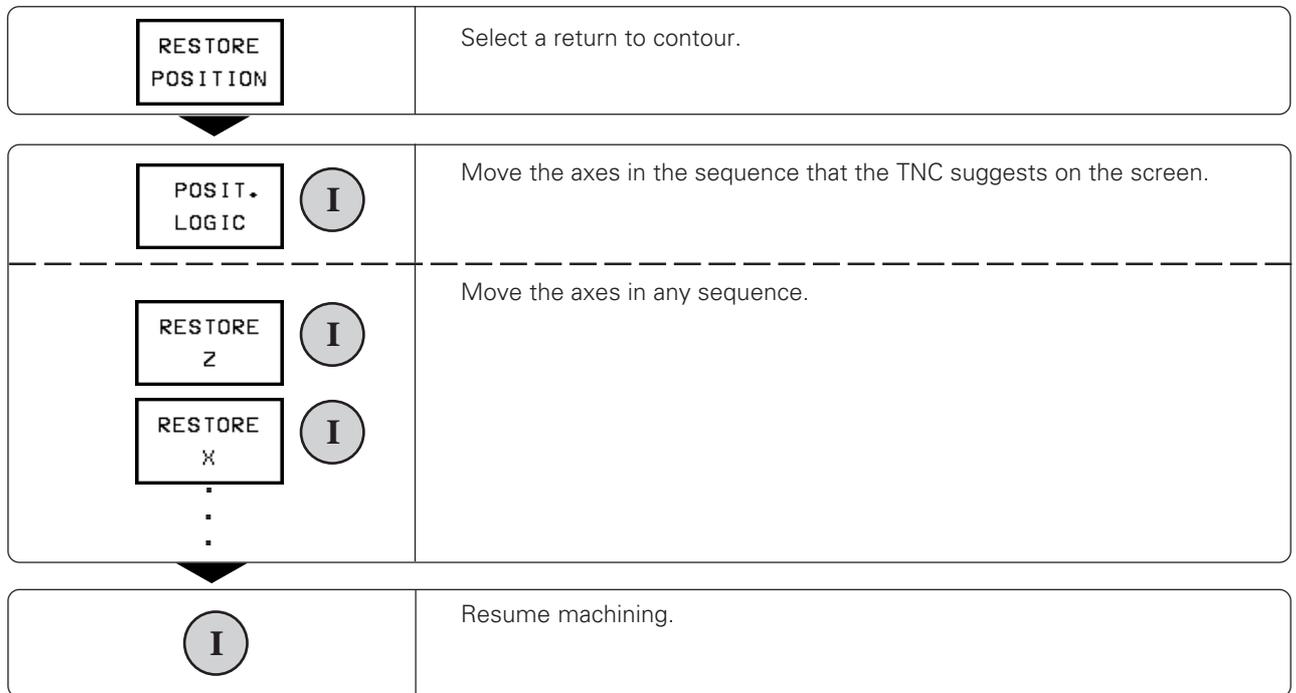
3.3 Optional Block Skip

In a test run or program run, the TNC can skip over blocks that you have programmed with a slash (/).

| ◁ or ▷ | Shift the soft-key row. |

PAGE ⇧	PAGE ⇩	BEGIN TEXT	END TEXT				╱▢ [OFF]╱ON

| ╱▢ OFF╱[ON] / ╱▢ [OFF]╱ON | Run or test the program with/without blocks preceded by a slash. |
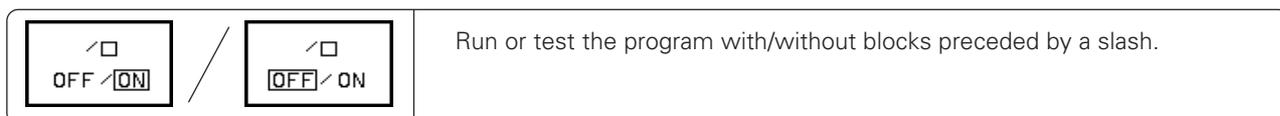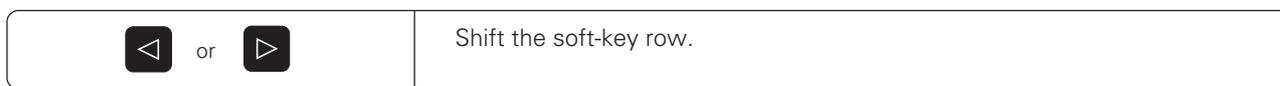
This function does not work with G99 blocks.

3.4 Blockwise Transfer: Testing and Running Long Programs

Programs that occupy more memory than the TNC provides can be "drip fed" block by block from an external storage device.

During program run, the TNC transfers program blocks from a floppy disk unit or PC through its data interface, and erases them after execution. This frees up memory for new blocks. (Coordinate transformations remain active even when the cycle definition has been deleted.)

To prepare for blockwise transfer:
• Prepare the data interface.
• Configure the data interface with the MOD function RS-232/422-SETUP (see page 10-4).
• If you wish to transfer a part program from a PC, interface the TNC and PC (see pages 9-5 and 11-3).
• Ensure that the transferred program meets the following requirements:
  - The highest block number must not exceed 99999999. The block numbers, however, can be repeated as often as necessary.
  - The program must not contain subprograms.
  - The program must not contain program section repeats.
  - All programs that are called from the transferred program must be selected (status M).

Fig. 3.1:   TNC screen during blockwise transfer

PROGRAM RUN / SINGLE BLOCK

or

PROGRAM RUN / FULL SEQUENCE

or

TEST RUN

EXT	Show directory of files in external storage. The soft-key row shifts.

Select the program.

SELECT	Start data transfer.

PROGRAM RUN: **I**	Execute the program blocks.
TEST RUN: START	Test the program blocks.
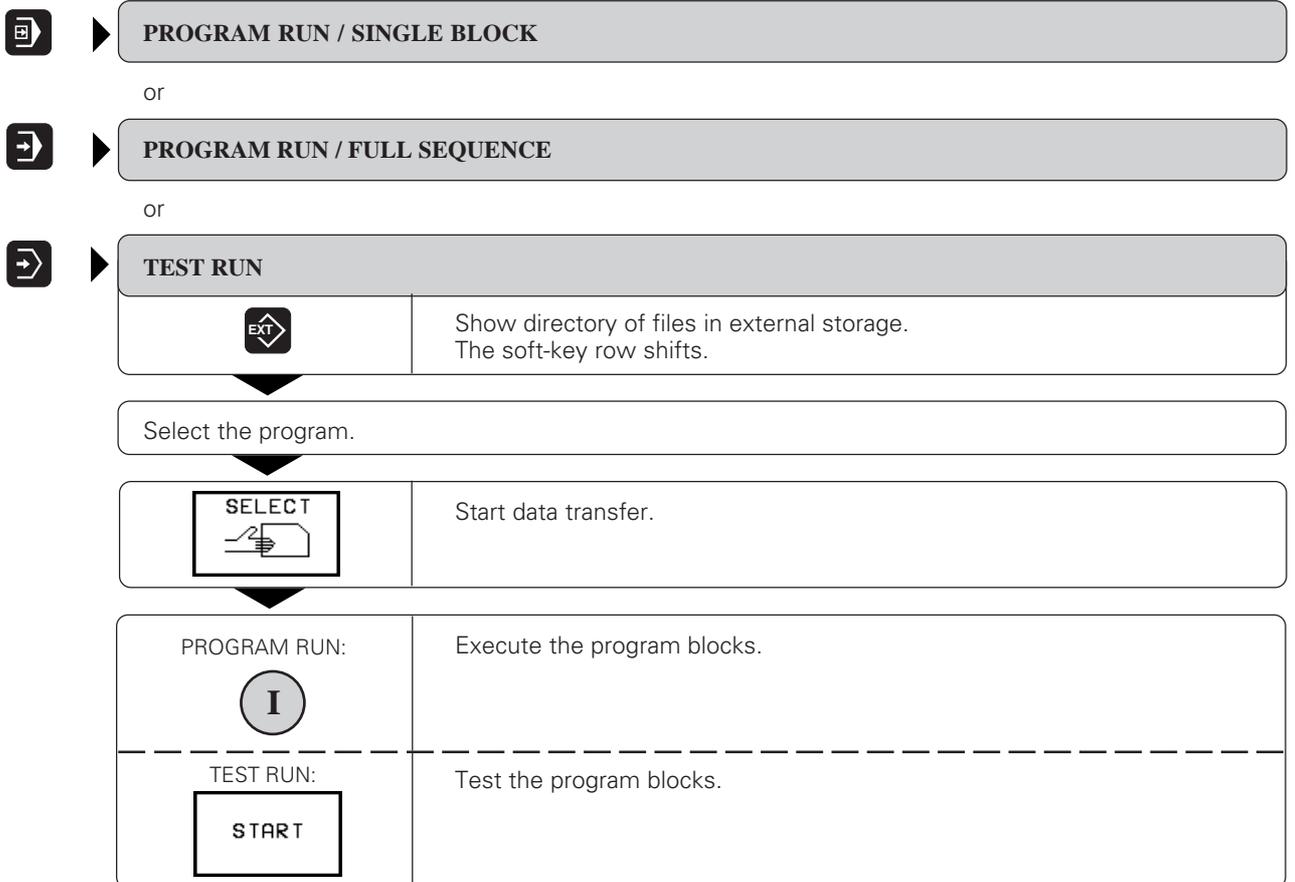
☞ If data transfer is interrupted, press the START key again.

3.4   Blockwise Transfer: Testing and Running Long Programs

Jumping over blocks

The TNC can jump over blocks to begin transfer at any desired block.
These blocks are then ignored during a program run or test run.

Select the program and start data transfer.	
GOTO e.g. **1 5 0** **ENT**	Go to the block number at which you wish to begin data transfer, for example 150.
PROGRAM RUN: **I**	Execute the transferred blocks, starting with the block number that you entered.
TEST RUN: **START**	Test the transferred blocks, starting with the block number that you entered.
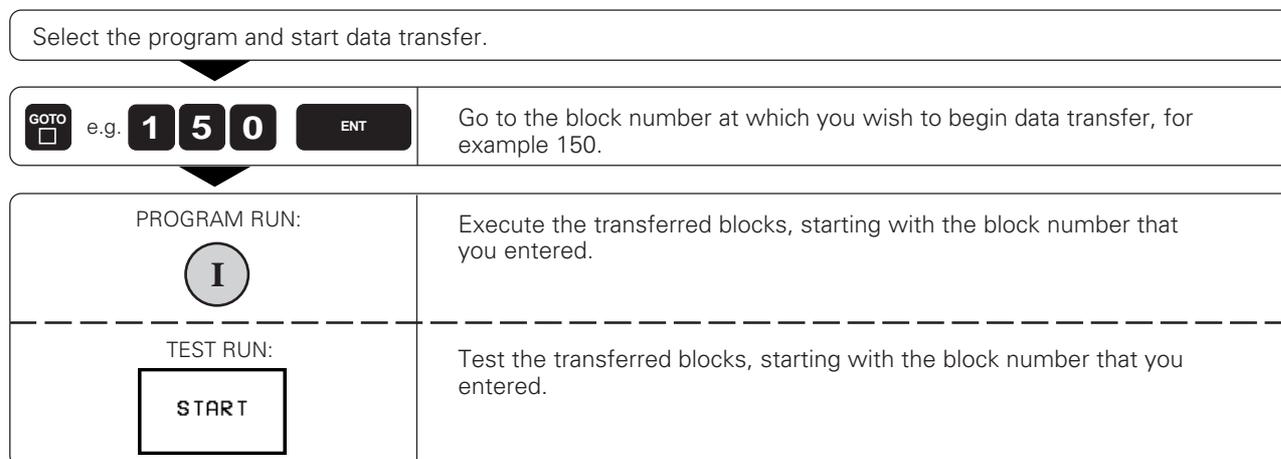
You can use machine parameter MP7228 (see page 11-12) to define the memory range to be used during blockwise transfer. This prevents the transferred program from filling the program memory and disabling the background programming feature.

As an alternative, you can call the external program with % EXT (see page 6-8) and perform a mid-program startup.

Example: To perform a mid-program start-up from block 12834 of external program GEH35K1 proceed as follows:

– Write the following short program:
   %START-UP G71
   N10 % EXT:GEH35K1
   N99999 %START-UP G71

– Select the START-UP program in the PROGRAM RUN/ FULL SEQUENCE mode of operation.

– Select the RESTORE POS AT N function and enter the desired block number, here 12834, for START-UP AT and the desired program, here GEH35K1, for PROGRAM.

– Start block scan with the NC START key.

4   Programming

In the PROGRAMMING AND EDITING mode of operation (see pages 1-30 and 1-40) you can

- create new files
- edit existing files

This chapter describes the basic functions and inputs that do not yet cause machine axis movement. The entry of geometry for workpiece machining is described in the next chapter.

4.1   Creating Part Programs

Layout of a program

A part program consists of individual program blocks. The TNC numbers the blocks in ascending sequence. The block number increment is defined in MP 7220 (see page 11-7). Program blocks consist of units of information called *words*.

Program block:

N10 G00   G40   G90   X+100 Y+20 M3

Path
function

Block
number

Words

Fig. 4.1:   Program blocks consist of words of specific information

Function	Key
Continue dialog	ENT
Ignore dialog question	NO ENT
End block	END
Delete block / delete word	DEL

Editing functions

Editing means entering, adding to or changing commands in programs.

The TNC enables you to

- Enter data with the keyboard
- Select desired blocks and words
- Insert and erase blocks and words
- Correct wrong values and commands
- Easily clear TNC messages from the screen

Types of inputs

Numbers, coordinate axes and radius compensation are entered directly by keyboard. You can set the algebraic sign either before, during or after a numerical entry.

Selecting blocks and words

- To call a block with a certain block number:

GOTO e.g. **1 0** **ENT**	The highlight jumps to block number 10.

- To move one block forwards or backwards:

↑ or **↓**	Press the vertical cursor keys.

- To select individual words in a block:

← or **→**	Press the horizontal cursor keys.

- To find the same word in other blocks:

← or **→**	Select the word in the block.
↑ or **↓**	Display the same word in other blocks.

Inserting blocks

- New program blocks can be inserted behind any existing block (except behind the N99999 block):

↑ or **↓** / **GOTO**	Select the block.
N e.g. **3 5** **ENT**	Program new block.

Editing and inserting words

Highlighted words can be changed as desired—simply overwrite the old value with the new one. After entering the new value, press a horizontal cursor key or the END key to confirm the change.

In addition to changing the existing words in a block, you can also add new words. Use the horizontal cursor keys to move the highlight to the block you wish to add words to.

Erasing blocks and words

Function	Key
Set the highlighted number to 0	CE
Erase an incorrect number	CE
Clear a non-blinking error message	CE
Delete the selected word	DEL
Delete the selected block	DEL
Erase program sections: First select the last block of the program section to be erased.	DEL

4.2 Tools

Each tool is identified by a number.

The tool data, consisting of the

- length L
- radius R

are assigned to the tool number.

The tool data can be entered

- into the individual part program in a G99 block, or
- once for each tool into a common tool table that is stored as a type .T file.

Once a tool is defined, the TNC associates its dimensions with the tool number and accounts for them when executing positioning blocks.

The way the tool is used is influenced by several miscellaneous functions (see page 11-16).

Setting the tool data

Tool numbers

Each tool is identified by a number between 0 and 254.

When the tool data are entered into the program, tool number 0 is automatically defined as having length L = 0 and radius R = 0. In tool tables, also, tool 0 should be defined with L = 0 and R = 0.

Tool radius R

The radius of the tool is entered directly.

Tool length L

The compensation value for the tool length is measured

- as the difference in length between the tool and a zero tool, or
- with a tool pre-setter.

A tool pre-setter eliminates the need to define a tool in terms of the difference between its length and that of another tool.

Oversizes for lengths and radii: delta values

In tool tables you can enter so-called delta values for tool length and radius.

- Positive delta values = tool oversize
- Negative delta values = tool undersize

Application

- Undersize in the tool table for wear

Delta values can be numerical values or 0.
The maximum permissible oversize or undersize is +/– 99.999 mm.
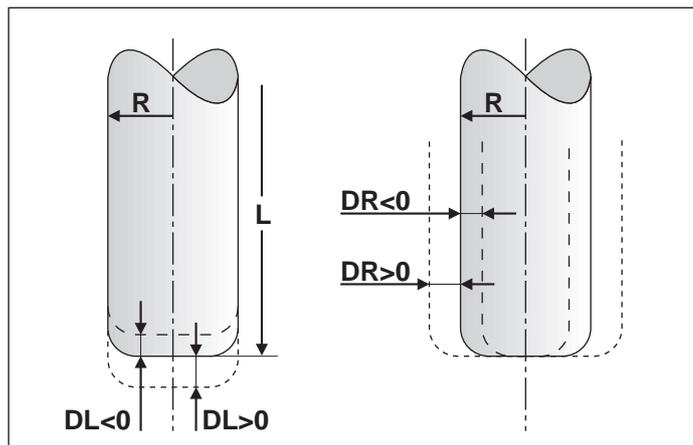
Fig. 4.2:   Oversizes DL, DR on a toroid cutter

Determining tool length with a zero tool

For the sign of the tool length L:

$L > L_0$ The tool is longer than the zero tool
$L < L_0$ The tool is shorter than the zero tool
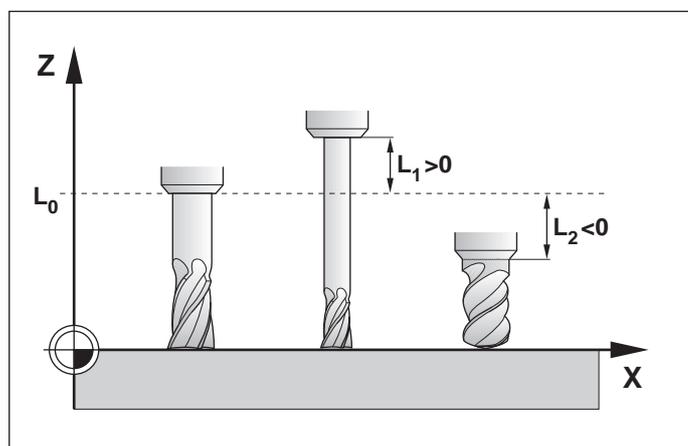
Fig. 4.3:   Tool lengths are entered as the difference from the zero tool

Move the zero tool to the reference position in the tool axis (e.g. workpiece surface with Z = 0).

If necessary, set the datum in the tool axis to 0.

Change tools.

Move the new tool to the same reference position as the zero tool.

The TNC displays the compensation value for the length L.

Note down the value and enter it later.

Enter the display value by using the "actual position capture" function (see page 4-26).

Entering tool data into the program

The following data can be entered once for each tool in the part program:

- Tool number
- Tool length compensation value L
- Tool radius R

To enter tool data into the program block:

| G | 9 | 9 | ENT | ▶ |

▶	**TOOL NUMBER ?**
e.g. **5** ENT	Give the tool a number, for example 5.

TOOL LENGTH L ?
e.g. **1 0** ENT	Enter the compensation value for the tool length, e.g. L = 10 mm.

TOOL RADIUS R ?
e.g. **5** ENT	Enter the tool radius, e.g. R = 5 mm.

Resulting NC block: G99 T5 L+10 R+5

You can enter the tool length L directly in the tool definition by using the "actual position capture" function (see page 4-26).

Entering tool data in tables

A tool table is a file containing the data for all tools. The maximum number of tools per table (0 to 254) is set in machine parameter MP7260.

On machines with automatic tool changers, the tool data must be stored in tool tables. You can edit these tool tables using special time-saving editing functions.

Types of tool tables

Tool table TOOL.T is
• used for machining
• edited in a program run mode of operation

All other tool tables are
• used for test runs and archiving
• edited in the PROGRAMMING AND EDITING mode of operation

> If you copy a tool table into TOOL.T for a program run, the old TOOL.T will be overwritten.

Editing functions for tool tables

The following functions help you to create and edit tool tables:

Function	Key / Soft key
Move the highlight vertically	↑ ↓
Move the highlight horizontally	← →
Go to the beginning of the table	BEGIN TABLE
Go to the end of the table	END TABLE
Go to the next table page	PAGE ⇩
Go to the previous table page	PAGE ⇧
Go to the beginning of the next line	NEXT LINE
Look for the tool name in the tool table	▷ FIND TOOL NAME
Display / Do not display the tool pocket number in the first column *(TNC 426 only)*	▷ SHOW OMIT TCHNR
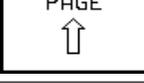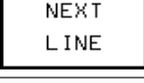
To edit the tool table TOOL.T:

	PROGRAM RUN / SINGLE BLOCK	

or

	PROGRAM RUN / FULL SEQUENCE	
TOOL TABLE	Select the tool table TOOL.T.	
EDIT OFF / ON	Switch the EDIT soft key to ON.	
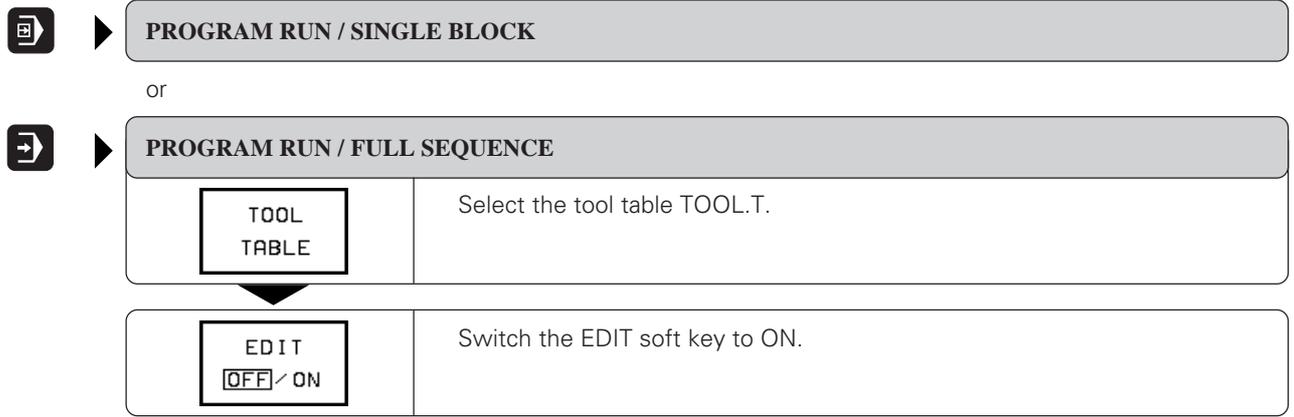
To edit a tool table other than TOOL.T:

	PROGRAMMING AND EDITING	
PGM NAME / PGM MGT	Call the file directory.	
SELECT TYPE SHOW .T	Shift the soft-key row and show file type .T.	

FILE NAME =   .T
Select the tool table.
Enter a new file name and create a new table.
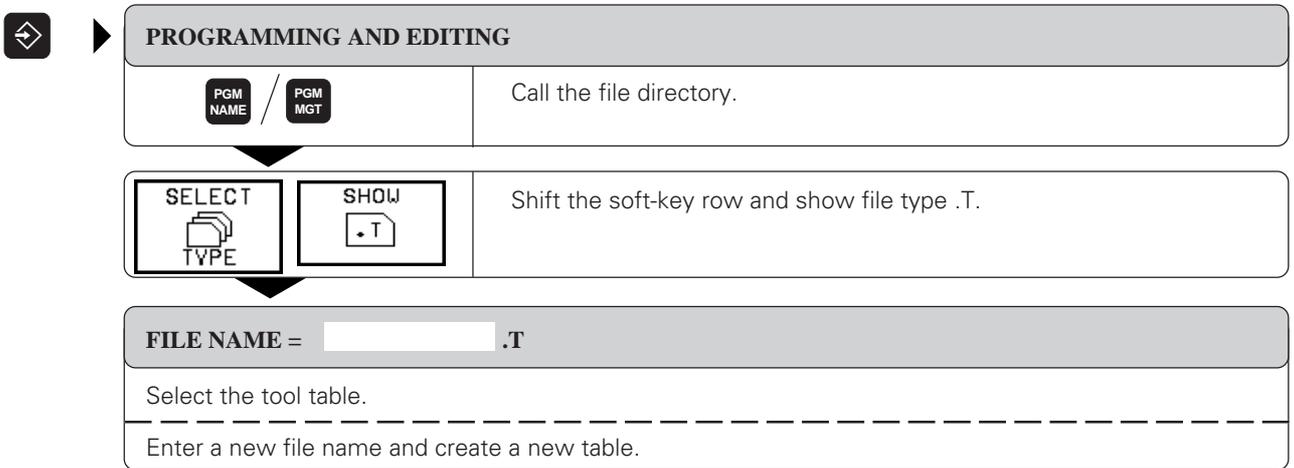
Tool data in tables

The following information can be entered in tool tables:

- Tool radius and tool length: R, L
- Curvature radius of the tool point for three-dimensional tool compensation: R2
  For graphic display of machining with a spherical cutter, enter R2 = R.
- Oversizes (delta values) for tool radii and tool lengths: DR, DR2, DL
- Tooth length of the tool: LCUTS
  (TNC 426 only)
- Maximum plunge angle of the tool: ANGLE
  (TNC 426 only)
- Tool name: NAME
- Maximum and current tool life: TIME1, TIME2, CUR.TIME
- Number of a replacement tool: RT
- Tool lock: TL
- Tool comment: DOC
- Information on this tool for the PLC (**p**rogrammable **l**ogical **c**ontrol: for interfacing the control to the machine): PLC

The following tool data only apply for automatic tool measurement (*conversational programming only*):

- Number of cutting edges for tool measurement: CUT
- Length tolerance for tool wear: LTOL
- Radius tolerance for tool wear: RTOL
- Cutting direction for dynamic tool measurement: DIRECT.
- Tool offset between stylus center and tool center: TT:R-OFFS
  Preset value: tool radius R
- Tool offset between upper edge of stylus and lower edge of tool: TT:L-OFFS
  Preset value: 0
- Length tolerance for tool break: LBREAK
- Radius tolerance for tool break: RBREAK

A general user parameter (MP7266) defines which data can be entered in the tool table and in what sequence the data is displayed.

The sequence of information in the tool table shown in the illustrations to the right is only one example out of many possibilities.

If all the information in a table no longer fits on one screen, this is indicated with >> or << in the line with the table name.

PROGRAM RUN FULL SEQUENCE	EDIT TOOL TABLE — TOOL RADIUS ?				
FILE: TOOLS		MM			>>
T	NAME	L	R	R2	DL	DR
0		+0	+0	+0	+0	+0
1	CUTTER1	+12.658	+3.75	+0.125	-0.68	-0.35
2	DRILLER	-2.59	+8.5	+0	+0	-0.07
3		+0	+0	+0	+0	+0
4		+0	+0	+0	+0	+0
5	CUTTER1A	+7.8	+3.5	+0.155	-0.06	-0.15
6		+0	+0	+0	+0	+0
7	BILLS	+1.25	+5	+0.5	+0.1	+0
8		+0	+0	+0	+0	+0
9		+12.59	+1	+0.05	+0.01	+0
10		+0	+0	+0	+0	+0
11		+0	+0	+0	+0	+0
12	CUTTER12	+12.658	+3.75	+0.125	-0.68	-0.35

| BEGIN TABLE | END TABLE | PAGE ⇩ | PAGE ⇧ | | NEXT LINE |

Fig. 4.4:   Left part of the tool table

PROGRAM RUN FULL SEQUENCE	EDIT TOOL TABLE — TOOL DESCRIPTION							
<< FILE: TOOLS			MM					
T	DL	DR	DR2	TL RT	TIME1	TIME2	CUR.TIME	DOC
0	+0	+0	+0		0	0	0	
1	-0.68	-0.35	+0.02	5	5800	5000	1256	FIRST TOOL
2	+0	-0.07	+0		4000	4600	1208	
3	+0	+0	+0		0	0	0	OLD TOOL
4	+0	+0	+0		0	0	0	
5	-0.06	-0.15	+0.015	1	6000	5800	2578	ROUGH
6	+0	+0	+0		0	0	0	
7	+0.1	+0	+0.01	L	1000	910	542	
8	+0	+0	+0		0	0	0	
9	+0.01	+0	+0.0015	L	800	650	125	POLISHING
10	+0	+0	+0		0	0	0	
11	+0	+0	+0		0	0	0	
12	-0.68	-0.35	+0.02	5	3000	2500	715	FINE WORKS

| BEGIN TABLE | END TABLE | PAGE ⇩ | PAGE ⇧ | | NEXT LINE |

Fig. 4.5:   Right part of the tool table

The following note applies to the TNC 426:
If you are copying a tool table from an external storage device into the TNC, you can overwrite individual lines or columns in the target table with the REPLACE FIELDS soft key. Prerequisites:
– The target table must exist.
– The file to be copied must only contain the lines or columns you want to replace.

To read out or read in a tool table (see page 9-4)**:**

⬥ EXT	Select external data input/output directly from the table.
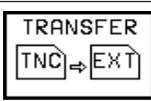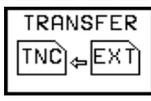
TRANSFER TNC ⇒ EXT	Read out the table.
TRANSFER TNC ⇐ EXT	Read in the table (only possible if EDIT ON is selected).

Abbreviation	Input	Dialog
T	Number by which the tool is called in the program	–
NAME	Name by which the tool is called in the program	TOOL NAME ?
L	Value for tool length compensation	TOOL LENGTH L ?
R	Tool radius R	TOOL RADIUS ?
R2	Tool radius R2 for toroid cutters (only for 3D radius compensation or graphical representation of a machining operation with spherical or toroid cutters)	TOOL RADIUS 2 ?
DL	Delta value for tool length	TOOL LENGTH OVERSIZE ?
DR	Delta value for tool radius R	TOOL RADIUS OVERSIZE ?
DR2	Delta value for tool radius R2	TOOL RADIUS OVERSIZE 2 ?
LCUTS	Tooth length of the tool: required for Cycle 22 *(TNC 426 only)*	TOOTH LENGTH IN TOOL AXIS ?
ANGLE	Maximum plunge angle of the tool for reciprocating plunge-cut *(TNC 426 only)*	MAXIMUM PLUNGE ANGLE ?
TL	Tool Lock	TOOL INHIBITED YES=ENT/NO=NOENT
RT	Number of a replacement tool, if available (see also TIME2	ALTERNATE TOOL ?
TIME1	Maximum tool life in minutes: The meaning of this information can vary depending on the individual machine tool. Your machine manual provides more information on TIME 1.	MAXIMUM TOOL LIFE ?
TIME2	Maximum tool life in minutes during TOOL CALL: If the current tool life exceeds this value, the TNC changes the tool during the next TOOL CALL (see also CUR.TIME).	MAXIMUM TOOL LIFE FOR TOOL CALL ?
CUR.TIME	Time in minutes that the tool has been in use: The TNC automatically counts the current tool life. A starting value can be entered for used tools.	CURRENT TOOL LIFE ?
DOC	Comment on tool (up to 16 characters)	TOOL DESCRIPTION ?
PLC	Information on this **tool** that is to be sent to the PLC	PLC STATUS?

Overview: Information in tool tables

Abbreviation	Input	Dialog
CUT.	Automatic tool measurement: number of cutting edges (20 cutting edges maximum)	NUMBER OF CUTTERS ?
LTOL	Automatic tool measurement: permissible deviation from tool length L for wear detection. If the entered value is exceeded, the TNC locks the tool (status L). Input range: 0 to 0.9999 mm	WEAR TOLERANCE: LENGTH ?
RTOL	Automatic tool measurement: permissible deviation from tool radius R for wear detection. If the entered value is exceeded, the TNC locks the tool (status L). Input range: 0 to 0.9999 mm	WEAR TOLERANCE: RADIUS ?
DIRECT.	Automatic tool measurement: cutting direction of the tool for dynamic tool measurement	CUTTING DIRECTION (M3 = –) ?
TT:R-OFFS	Automatic tool length measurement: tool offset between stylus center and tool center. Preset value: tool radius R	TOOL OFFSET: RADIUS ?
TT:L-OFFS	Automatic tool radius measurement: tool offset in addition to MP 6530 (see page 11-6) between upper edge of stylus and lower edge of tool. Preset value: 0	TOOL OFFSET: LENGTH ?
LBREAK	Automatic tool measurement: permissible deviation from tool length L for break detection. If the entered value is exceeded, the TNC locks the tool (status L). Input range: 0 to 0.9999 mm	BREAK TOLERANCE: LENGTH ?
RBREAK	Automatic tool measurement: permissible deviation from tool radius R for break detection. If the entered value is exceeded, the TNC locks the tool (status L). Input range: 0 to 0.9999 mm	BREAK TOLERANCE: RADIUS ?

Overview: Information in tool tables

Pocket table for tool changer

The **TOOL_P** table (for tool pocket) is programmed in a program run operating mode.

The soft key NEW POCKET TABLE (or RESET POCKET TABLE) is for erasing an existing pocket table and writing a new one.

Like the tool table, a pocket table can also be read in and read out directly through the data interface (see page 4-11).

```
EDIT TOOL TABLE                         EDIT
POCKET LOCKED YES=ENT/NO=NOENT          TOOL TABLE
  FILE: TOOL_P
P    T   ST F  L  PLC
0    0      F     %11010011
1    2   S     ▪  %11010001
2              L  %00000000
3    12           %11011001
4              L  %00000000
5    3   S  F     %11010011
6              L  %11011011


ACTL.       X     +12.759      Y       -5.370
            Z    +105.000      U      +45.001
            W    -230.987
T           ▨ ▨              F  0       M 5/9
 BEGIN   END    PAGE   PAGE   RESET   EDIT   NEXT   TOOL
 TABLE   TABLE   ⇩      ⇧    POCKET  OFF/ON  LINE  TABLE
                             TABLE
```

Fig. 4.6:   Pocket table for the tool changer

To select the pocket table:

TOOL TABLE	Select tool table.
POCKET TABLE	Select pocket table.
EDIT OFF/ON	Set the EDIT soft key to ON.
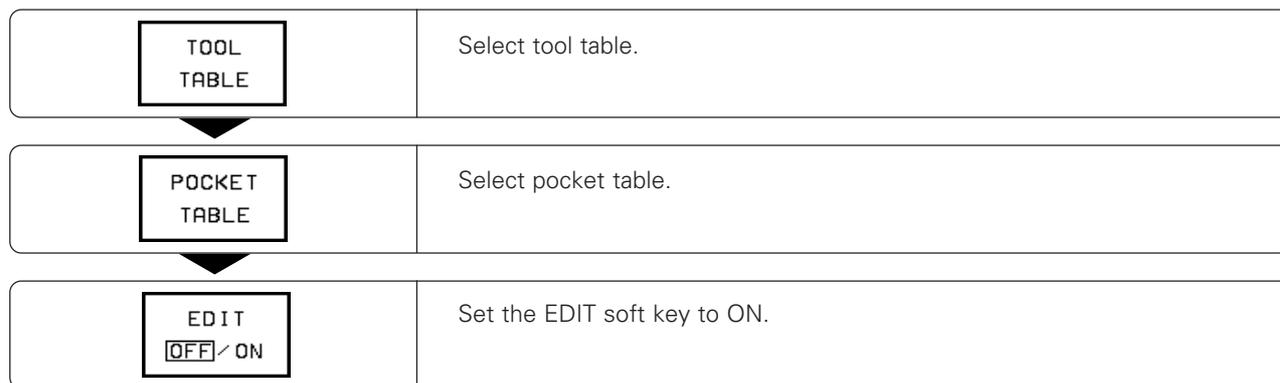
To edit the pocket table:

Abbreviation	Input	Dialog
P	Pocket number of the tool	–
T	Tool number	TOOL NUMBER
F	Fixed tool number. The tool is always returned to the same pocket.	FIXED POCKET YES = ENT / NO = NOENT
L	Locked pocket	POCKET LOCKED YES = ENT / NO = NOENT
ST	Special tool with a large radius requiring several pockets in the tool magazine. If your special tool takes up pockets in front of and behind its actual pocket, these additional pockets need to be locked.	SPECIAL TOOL
PLC	Information on this **tool pocket** that is to be sent to the PLC	PLC STATUS

Overview: Information in pocket tables

Calling tool data

The following data can be programmed in the NC block with T:

• Tool number, Q parameter
• Working plane with G17/G18 or G19
• Spindle speed S

To call tool data:

T ▶	**TOOL NUMBER ?**	
	e.g. **5**	Enter the number of the tool as defined in the tool table or in a G99 block, for example 5.
	G 1 7	Select the spindle axis Z.
	S 5 0 0 END	Enter the spindle speed, e.g. S=500 rpm.
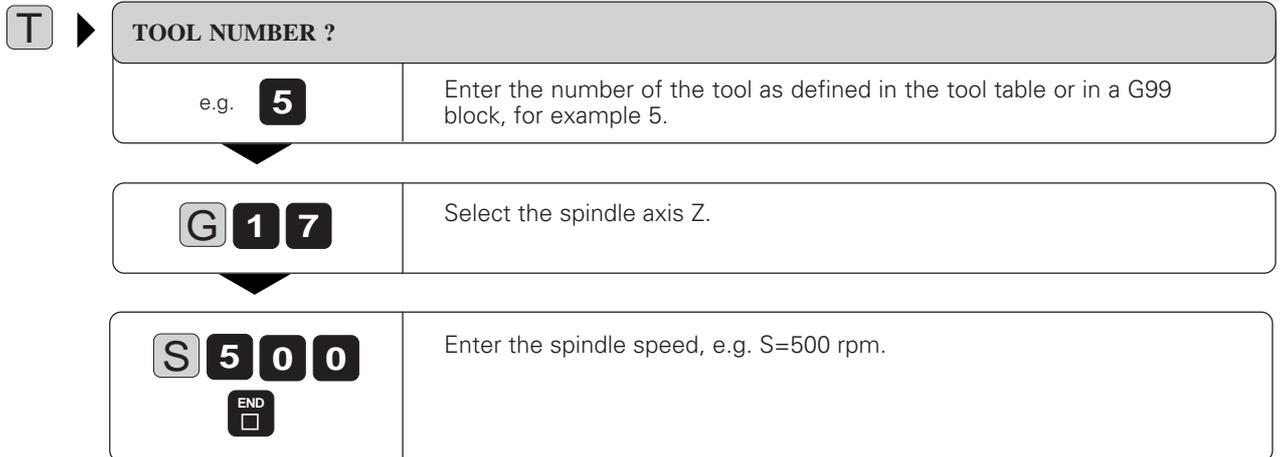
Resulting NC block: T5 G17 S500

Tool pre-selection with tool tables

If you are using tool tables, G51 pre-selects the next tool. Enter the tool number or a corresponding Q parameter.

Tool change

> The tool change function can vary depending on the individual machine tool. Your machine manual provides more information on this function.

Automatic tool change

If your machine has automatic tool changing capability, the TNC controls the replacement of the inserted tool by another from the tool magazine. The program run is not interrupted.

Manual tool change

To change the tool manually, stop the spindle and move the tool to the tool change position. Sequence of action:

• Move to the tool change position (under program control, if desired)
• Interrupt program run (see page 3-5)
• Change the tool
• Resume the program run (see page 3-6)

Tool change position

A tool change position must be located next to or above the workpiece where no collisions are possible. With the miscellaneous functions M91 and M92 (see page 5-39) you can enter machine-referenced (rather than workpiece-referenced) coordinates for the tool change position.

If T0 is programmed before the first tool call, the TNC moves the tool spindle in the tool axis to a position that is independent of the tool length.

If a positive length compensation was in effect before T0, the clearance to the workpiece will be reduced.

Automatic tool change: M101

This function can vary depending on the individual machine tool. Your machine manual provides more information on M101.

Standard behavior – without M101

When the tool reaches the maximum tool life (TIME1), the TNC flags this information in the TNC memory. The machine tool builder determines how the individual machine tool will respond to this condition (refer to the machine manual).

Automatic tool change – with M101

The TNC automatically changes the tool if the tool life (TIME1 or TIME2) expires during program run. However, the tool is not changed immediately after expiration of the tool life, but, depending on the workload of the processor, a few NC blocks later.

Duration of effect

M101 is reset with M102.

Standard NC blocks with radius compensation G40, G41, G42

The radius of the replacement tool must be the same as that of the original tool. If the radii are not equal, the TNC displays an error message and does not replace the tool.

4.3 Tool Compensation Values

For each tool, the TNC offsets the spindle path in the tool axis by the compensation value for the tool length and in the working plane by the compensation value for the tool radius.
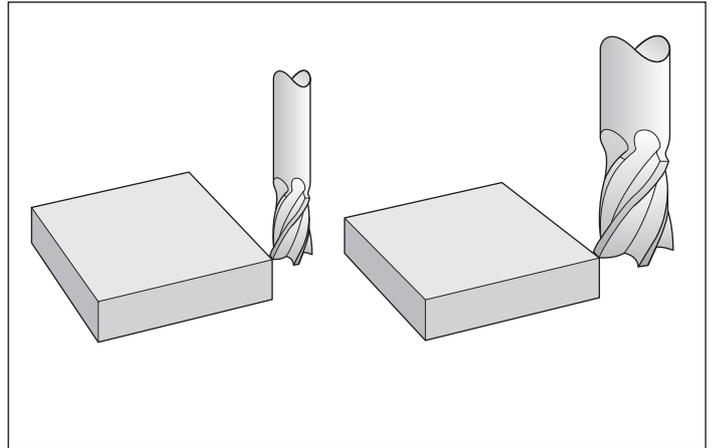
Fig. 4.7:   The TNC compensates both the length and radius of the tool

Effect of tool compensation values

Tool length

Length compensation becomes effective automatically as soon as a tool is called and the tool axis moves.

To cancel length compensation, call a tool with length L = 0.

If a positive length compensation was active before tool T0 was called, the distance to the workpiece will be reduced. With a G91 movement in the tool axis after a tool call with T, the length difference between the previous tool and the new tool will be traversed in addition to the programmed value.

Tool radius

Radius compensation becomes effective as soon as a tool is called and is moved in the working plane with G41 or G42.

To cancel radius compensation, program a positioning block with G40.

Tool radius compensation

A tool movement can be programmed:

- Without radius compensation (G40)
- With radius compensation (G41 or G42)
- As paraxial movements (G43 or G44)
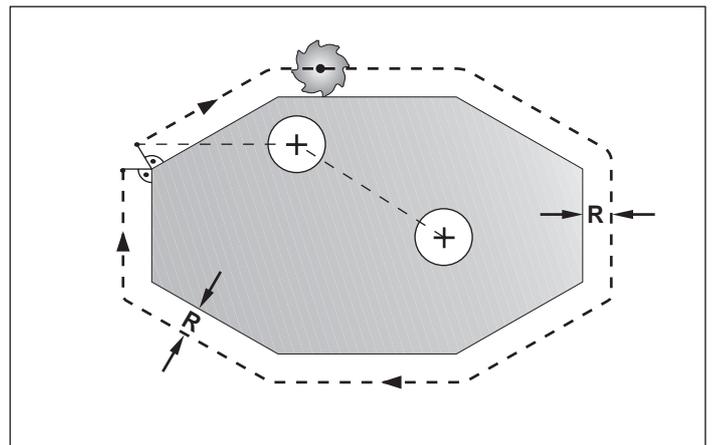
Fig. 4.8:   Programmed contour (— , +) and the path of the tool center (– – –)

Movement without radius compensation: G40

The tool center moves to the programmed coordinates.

Applications:

* Drilling and boring
* Pre-positioning
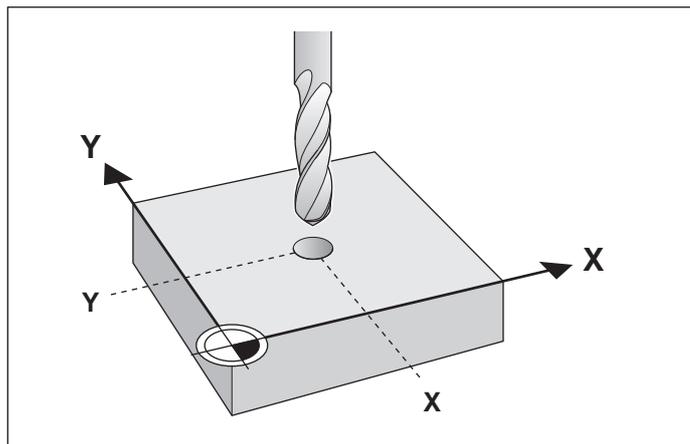
Fig. 4.9:   These drilling positions are entered without radius compensation

Tool movement with radius compensation: G41, G42

* Between two program blocks with different radius compensations you must program at least one block without radius compensation (that is, with G40).
* Radius compensation does not come into effect until the end of the block in which it is first programmed.
* Whenever radius compensation is activated or cancelled, the TNC positions the tool perpendicular to the programmed starting or end position. Position the tool at a sufficient distance from the first (or last) contour point to prevent the possibility of damaging the contour.

The tool center moves to the left (G41) or right (G42) of the programmed contour at a distance equal to the radius. "Left" and "right" are to be understood as based on the direction of tool movement, assuming a stationary workpiece.
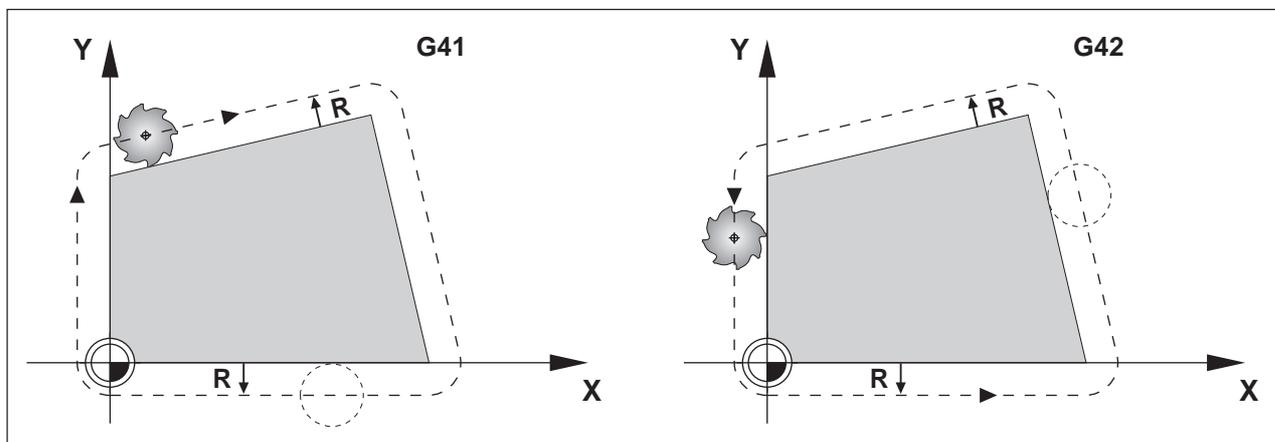
Fig. 4.10:   The tool moves to the left *(G41)* or right *(G42)* of the path during milling

Shortening or lengthening single-axis movements: G43, G44

This type of radius compensation is only possible for single-axis movements in the working plane. The programmed tool path is lengthened (G43) or shortened (G44) by the tool radius.

Applications:

* Single-axis machining
* Occasionally for pre-positioning the tool, such as for cycle G47 SLOT MILLING.

* You can enable G43 and G44 by programming a positioning block with an axis key.
* The machine tool builder can set machine parameters to inhibit programming of single-axis positioning blocks.

Machining corners

👆 If you are working without radius compensation, you can influence the machining of outside corners with M90 (see page 5-36).

Outside corners

The TNC moves the tool in a transitional arc around outside corners. The tool "rolls around" the corner point.

If necessary, the feed rate F is automatically reduced at outside corners to reduce stress on the machine, for example with very great changes in direction.
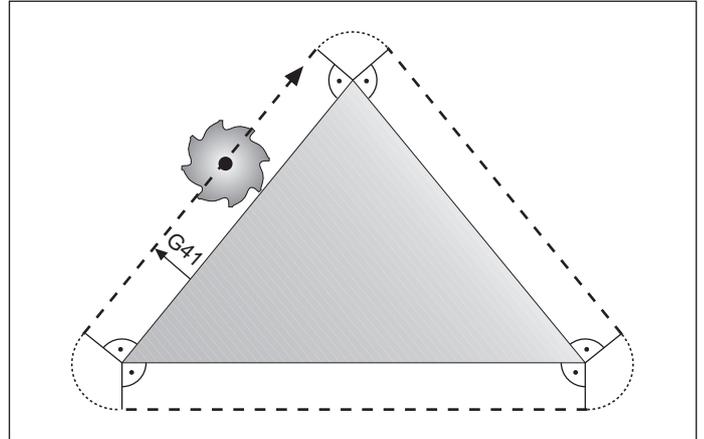
Fig. 4.11:   The tool "rolls around" outside corners

Inside corners

👆 To prevent the tool from damaging the contour, be careful not to program the starting (or end) positions for machining inside corners at a corner of the contour.

The TNC calculates the intersection of the tool center paths at inside corners. From this point it then starts the next contour element. This prevents damage to the workpiece.

The permissible tool radius, therefore, is limited by the geometry of the programmed contour.
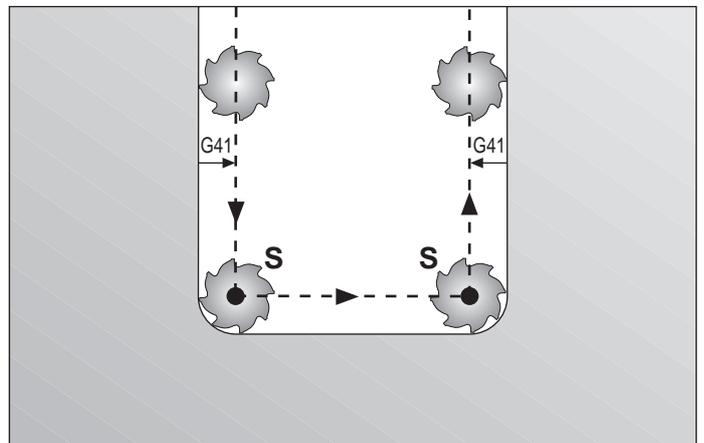
Fig. 4.12:   Tool path for inside corners

4.4 Program Creation

Defining the blank form

If you wish to use the TNC's graphic workpiece simulation you must first define a rectangular workpiece blank. Its sides lie parallel to the X, Y and Z axes and can be up to 30,000 millimeters long.
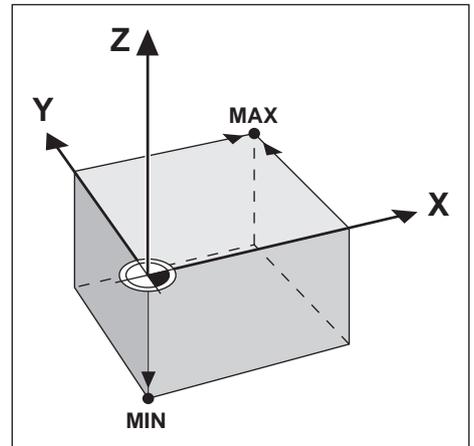
Fig. 4.13:   MIN and MAX points define the blank form.

The ratio of the blank-form side lengths must be less than 200:1.

MIN and MAX points

The blank form is defined by two of its corner points:

- MIN point: smallest X, Y and Z coordinates of the blank form, entered as absolute values.
- MAX point: largest X, Y and Z coordinates of the blank form, entered as absolute or incremental values.

To create a new part program:

PGM NAME / PGM MGT	Select the file management.

Select any file of type .I (for example OLD .I).

FILE NAME = OLD .I

e.g. N E W ENT	Enter the name of the new file (for example NEW .I).

TNC 407/TNC 415B/TNC 425:   If the dimensions in the program will be entered in millimeters (G71), press the ENT key.

If the dimensions in the program will be entered in inches (G70), press the NOENT key.

TNC 426:   If the dimensions in the program will be entered in millimeters (G71), press the MM soft key.

If the dimensions in the program will be entered in inches (G70), press the INCH soft key.

G 3 0	G function for input of the MIN point.

G 1 7	Define the tool axis: G17 means the Z axis.
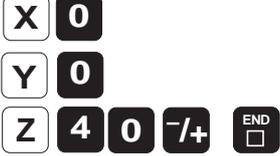
e.g. X 0 Y 0 Z 4 0 -/+ END	Enter, in sequence, the X, Y and Z coordinates of the MIN points, and conclude the block with END.

G 3 1	G function for input of the MAX point.

G 9 0	Entry as absolute value, or
G 9 1	as incremental value.

e.g. X 1 0 0 Y 1 0 0 Z 0 END	Enter, in sequence, the X, Y and Z coordinates of the MAX point, and conclude the block with END.

The following blocks then appear on the TNC screen as program text:

> **% NEW G71 ✳**

Block 1: Program begin, name, dimensional unit

> **N10 G30 G17 X+0 Y+0 Z–40 ✳**

Block 2: Tool axis, MIN point coordinates

> **N20 G31 G90 X+100 Y+100 Z+0 ✳**

Block 3: MAX point coordinates

> **N99999 % NEW G71 ✳**

Block 4: Program end, name, dimensional unit

The dimensional unit used in the program appears behind the program name (G71 = millimeters).

4.5 Entering Tool-Related Data

Besides the tool data and compensation, you must
also enter the following information:

• Feed rate F
• Spindle speed S
• Miscellaneous functions M
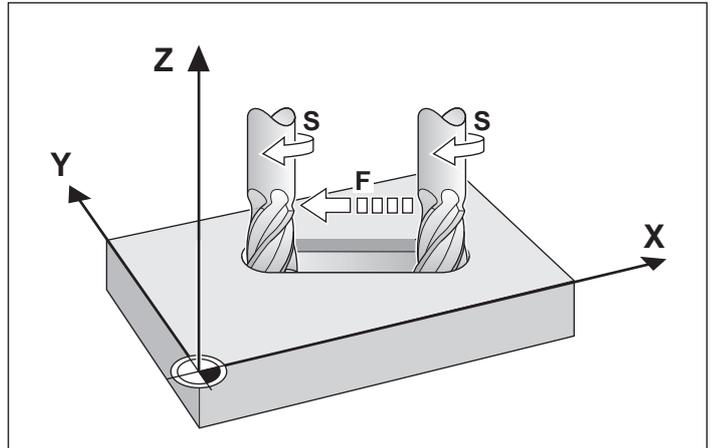
Fig. 4.14:   Feed rate F and spindle speed S of the tool

Feed rate F

The feed rate is the speed (in millimeters per minute or inches per
minute) at which the tool center moves.

Input range:
F = 0 to 300,000 mm/min (1181 ipm)

The maximum feed rate is set individually for each axis by means of
machine parameters.

Input

F ▶	e.g. **1 0 0**	Enter the feed rate, for example F = 100 mm/min.

Rapid traverse

Rapid traverse is programmed directly with G00.

Duration of feed rate F

A feed rate entered as a numerical value remains in effect until the control
encounters a block with a different feed rate.

If the new feed rate is G00 (rapid traverse), then after the next block with
G01 the feed rate will return to the last feed rate entered as a numerical
value.

Changing the feed rate F

You can adjust the feed rate with the override knob on the TNC keyboard
(see page 2-6).

Spindle speed S

The spindle speed S is entered in revolutions per minute (rpm).

Input range:
S = 0 to 99,999 rpm

To change the spindle speed S in the part program:

| S ▶ | e.g. **1 0 0 0**
 END □ | Enter the spindle speed S, for example 1000 rpm. |

Resulting NC block: T1 G17 S1000

To adjust the spindle speed S during program run:

| 100
 50 ⊕ 150
 0 ↻ S % | On machines with stepless spindle drives, the spindle speed S can be varied with the override knob. |

4.6 Entering Miscellaneous Functions and Program Stop

Some M functions are not effective on certain machines. The machine tool builder may also add some of his own M functions. Refer to your machine manual for more information.

The M functions (M for miscellaneous) affect:

• Program run
• Machine functions
• Tool behavior

The back cover foldout of this manual contains a list of M functions that are predetermined for the TNC. The list indicates whether an M function becomes effective at the start or at the end of the block in which it is programmed.

An NC block can contain several M functions as long as they are independent of each other. Refer to the overview on the last cover page to see how the M functions are grouped.

A program run or test run will be interrupted when it reaches a block containing G38.

If you wish to interrupt the program run or test run for a certain length of time, use the Cycle G04: DWELL TIME (see page 8-53).

4.7 Actual Position Capture

Sometimes you may want to enter the actual position of the tool in a particular axis as a coordinate in a part program. Instead of reading the actual position values and entering them with the numeric keypad, you can simply press the "actual position capture" key (see illustration at right). You can use this feature to enter, for example, the tool length.
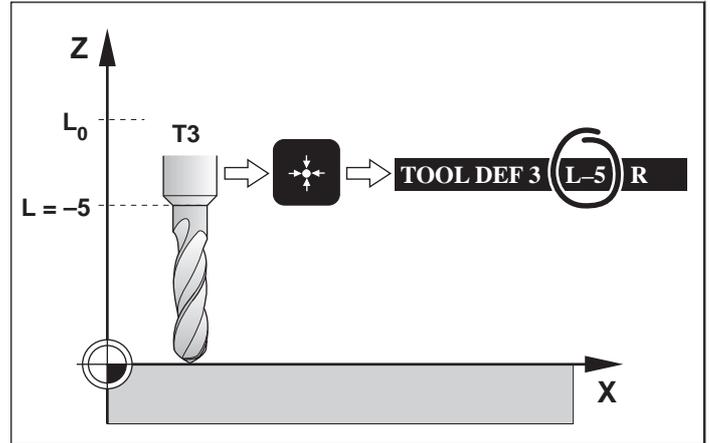
Fig. 4.15:   Storing the actual position in the TNC

To capture the actual position:

	MANUAL OPERATION
	Move the tool to the position that you wish to capture.

	PROGRAMMING AND EDITING
	Select or create the program block in which you wish to enter the actual position of the tool.

e.g. **X**	Select the axis in which you wish to capture a coordinate, for example X.

	Transfer the actual position coordinate to the program.

Enter the radius compensation according to the position of the tool relative to the workpiece.

4.8 Integrated Pocket Calculator

With the CALC key you can open an additional window for performing calculations.

The calculator is operated with short commands through the ASCII keyboard. The commands are shown in a special color in the calculator window. The calculator can perform the following mathematical operations:
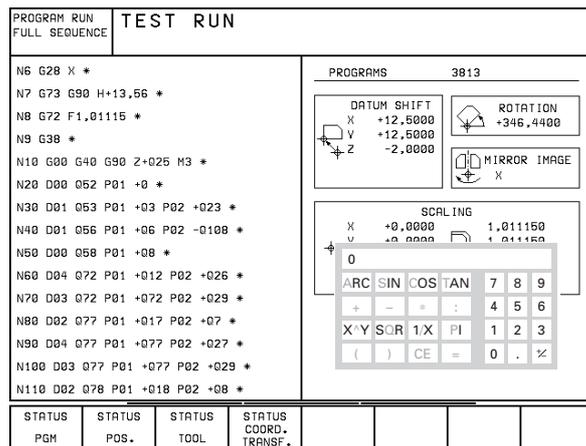
Fig. 4.16:   The TNC's pocket calculator

Mathematical function	Command
Addition	+
Subtraction	–
Multiplication	✳
Division	:
Sine	S
Cosine	C
Tangent	T
Arc sine	AS
Arc cosine	AC
Arc tangent	AT
Powers	^
Square root	Q
Inversion	/
Parenthetic calculations	( )
π (3.14159265359)	P
Display result	=

To move the pocket calculator:

➤ Press the arrow keys.

To close the pocket calculator:

➤ Press the CALC key again.

Example:

Find the sine of 30° and multiply by 50.

➤ Press the CALC key.
➤ Enter the number 30.
➤ Press the S key (for sine) on the ASCII keyboard.
➤ Press the * on the ASCII keyboard.
➤ Enter the number 50.
➤ Press the = key on the ASCII keyboard, the TNC displays the result (25) at the top of the calculator. You can use the "actual position capture" key to transfer the result to the highlight position in the current block (only available in the PROGRAMMING AND EDITING mode of operation).

4.9 Marking Blocks for Optional Block Skip

You can mark program blocks so that the TNC will skip them during a program or test run whenever the block skip option is active (see page 3-10).

To mark a block:

Select the desired block.	

/	Mark the beginning of the block with a slash.

- Blocks containing a tool definition (G99) cannot be skipped.
- To skip cycles, program the slash in the first block of the cycle.

To erase the slash:

Select the block that contains the slash to be erased.	

x	Erase the slash at the beginning of the block.

4.10 Text Files

You can use the TNC's text editor to write and edit texts.

Typical applications:

- Recording test results
- Documenting working procedures
- Keeping formulas and creating cutting data diagrams

The text editor can edit only type .A files (text files). If you want to edit other types of files with the text editor, you must first convert them (see pages 1-39 and 1-43).

The typewriter-style keyboard provides letters, symbols and function keys (e.g., backspace) that you need to create and change texts. The soft keys enable you to move around in the text and to find, delete, copy and insert letters, words, sections of text (text blocks), or entire files.

To create a text file:

PGM NAME	▶	**PROGRAMMING AND EDITING**	
PGM MGT		SELECT TYPE and SHOW .A	Show text files (type .A files).

FILE NAME = _____ **.A**	
e.g. A B C ENT	Enter a file name, for example ABC, and confirm.
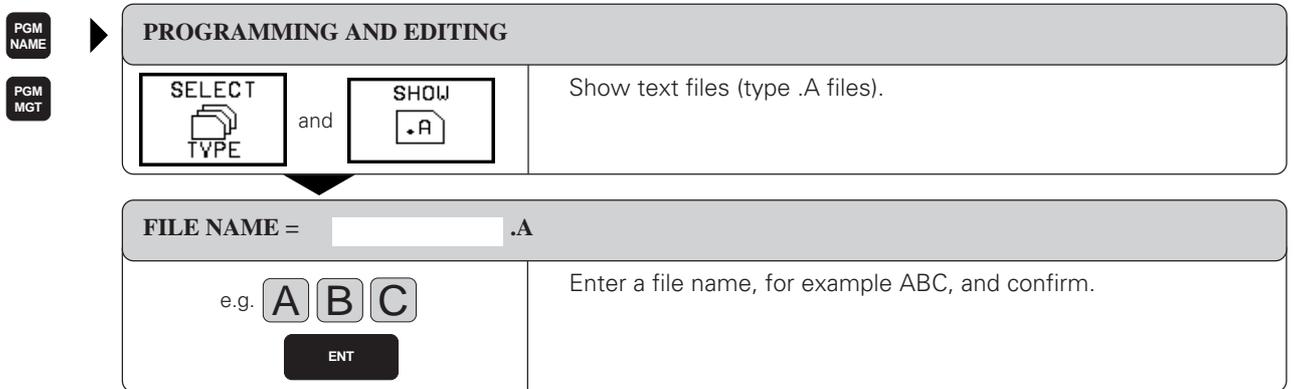
The following information is visible in the high-lighted line at the top of the text window:

- FILE: Name of the current text file
- LINE: Line in which the cursor is presently located
- COLUMN: Column in which the cursor is presently located
- INSERT: Insert new text, pushing the existing text to the right
- OVERWRITE: Write over the existing text, erasing it where it is replaced with the new text.

You can toggle between the INSERT and OVER-WRITE modes with the soft key at the far left. The selected mode is shown enclosed in a frame.


```
PROGRAM RUN   PROGRAMMING AND EDITING
SINGLE BLOCK

FILE: TEXTFILE        LINE: 13   COLUMN: 1   INSERT

THIS IS A TEXT FILE...

IN THE TEXT FILE YOU MAY

- RECORD TEST RESULTS
- DOCUMENT WORKING PROCEDURES
- STORE FORMULAS AND TABLES
- WRITE MESSAGES
- RECORD MACHINE PARAMETERS

ETC.

■
[END]

[INSERT]   MOVE    MOVE    PAGE    PAGE    BEGIN   END    FIND
OVERWRITE  WORD    WORD     ⇩       ⇧     TEXT   TEXT
           >>      <<
```
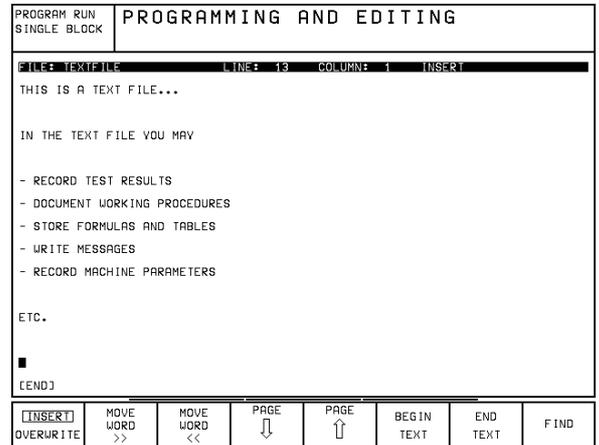
Fig. 4.17:   TNC text editor screen

To leave a text file:

PGM NAME	▶	SELECT TYPE SHOW .I	Select a different type of file, for example programs in ISO format.
PGM MGT			

	↑ ↓ + SELECT	Choose the desired program.
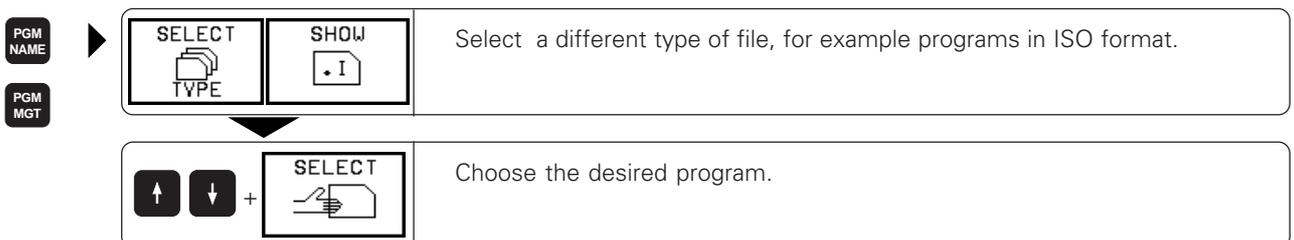
The text that you type always appears on the screen where the cursor is located. You can move the cursor with the cursor keys and the following soft keys:

Function	Soft key
Move one word to the right	MOVE WORD >>
Move one word to the left	MOVE WORD <<
Go to the next screen page	PAGE ⇩
Go to the previous screen page	PAGE ⇧
Go to beginning of file	BEGIN TEXT
Go to end of file	END TEXT

In each screen line you can enter up to 77 characters from the alphabetic and numeric keypads.

The alphabetic keyboard offers the following function keys for editing text:

Function	Key
Begin a new line	RET
Erase character to left of cursor (backspace)	⟨X⟩
Insert a blank space	SPACE

Exercise:

Write the following text in the file ABC.A. You will need it for the exercises in the next few pages.

JOBS
!! IMPORTANT:

MACHINE THE CAMS (ASK THE BOSS?!)
PROGRAM 1375.H; 80% OK
BY LUNCH

TOOLS
TOOL 1 DO NOT USE
TOOL 2 CHECK
REPLACEMENT TOOL: TOOL 3
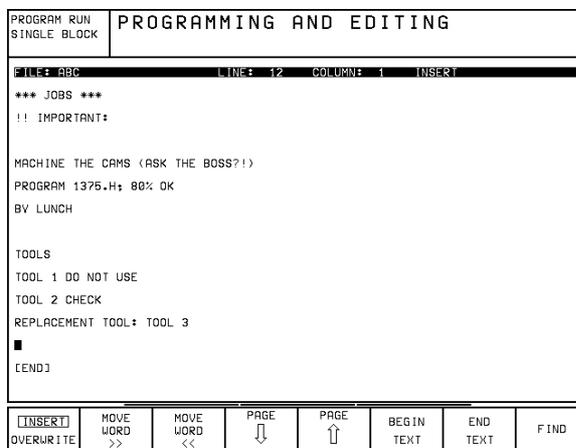
Fig. 4.18:   Text editor screen with exercise text

Finding text sections

You can search for a desired character or word with FIND at the far right
of the first soft-key row. The following functions then appear:

FIND CURRENT WORD						EXECUTE	END

Finding the current word

You can search for the next occurrence of the word in which the cursor is
presently located.

Exercise: Find the word TOOL in the file ABC.A

Move the cursor to the word TOOL.	
FIND	Select the search function.

FIND TEXT :   TOOL

FIND CURRENT WORD	Search for the current word (TOOL).

To find any text:

FIND	Select the search function.

FIND TEXT :

Enter the text that you wish to find.
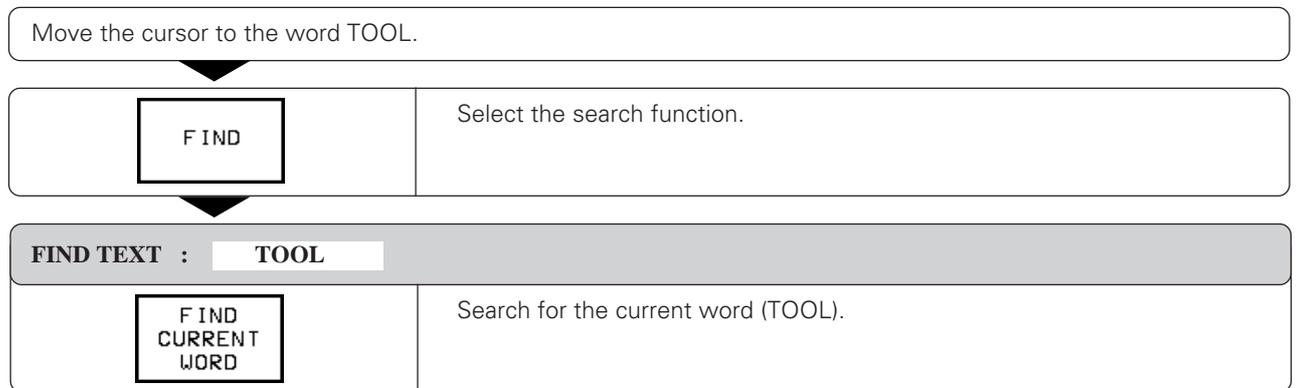
EXECUTE	Find the text.
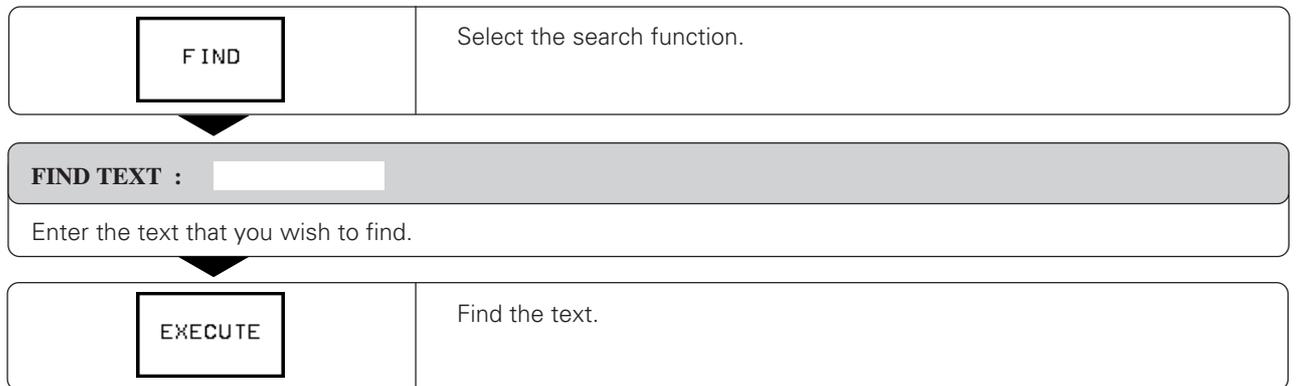
To leave the search function:

END	Terminate the search function.
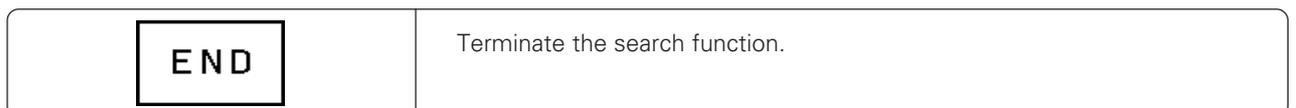
To erase and insert characters, words and lines:

| ◁ or ▷ | Shift the soft-key row. |

DELETE CHAR	DELETE WORD	DELETE LINE	RESTORE LINE/WORD				
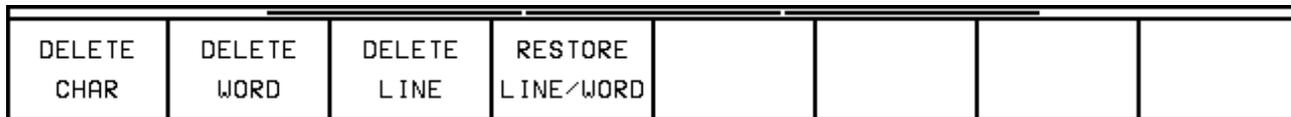
Move the cursor to the text that you wish to erase, or to the place where you wish to insert text.

Function	Soft key
Delete a character	DELETE CHAR
Delete and temporarily store a word	DELETE WORD
Delete and temporarily store a line	DELETE LINE
Insert a line/word from temporary storage	RESTORE LINE/WORD

Exercise: Delete the first line of ABC.A and insert it behind BY LUNCH

	Move the cursor to any position in the line *** JOBS ***.
▷	Shift the soft-key row.
DELETE LINE	Delete the line and store temporarily.
↓	Move the cursor to the beginning of the line behind BY LUNCH.
RESTORE LINE/WORD	Insert the line *** JOBS *** at the cursor position.
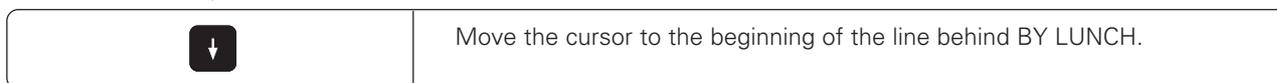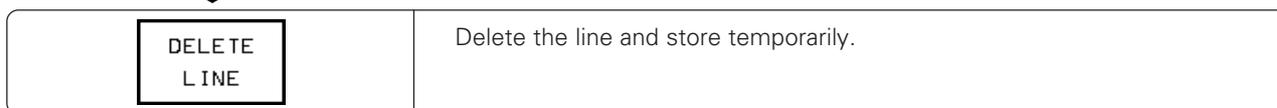
Temporarily stored words and lines can be inserted as often as desired.

Editing text blocks

With the editor, text blocks (sections of text) of any size can be

• selected
• deleted
• inserted at the same or other locations
• copied (even whole files)
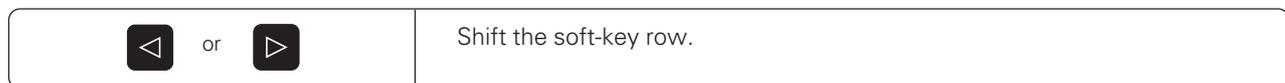
| ◁ or ▷ | Shift the soft-key row. |

| SELECT BLOCK | REMOVE BLOCK | INSERT BLOCK | REMOVE/ INSERT BLOCK | | | APPEND TO FILE | READ FILE |

Function	Soft key
To select a block: Place the cursor at one end of the block and press SELECT BLOCK. Then move the cursor to the other end. The selected block has a different color than the rest of the text.	SELECT BLOCK
Delete the selected text and store temporarily	REMOVE BLOCK
Insert the temporarily stored text at the cursor location	INSERT BLOCK
Store marked block temporarily without erasing	REMOVE/ INSERT BLOCK
Transfer the selected text to another file: Type the name of the target file in the screen dialog line and press ENT. The TNC appends the selected text to the end of the specified file. You can also create a new file with the selected text in this way.	APPEND TO FILE
Insert another file at the cursor position: Write the name of the source file in the screen dialog line and press ENT.	READ FILE

Exercise:

Move the last four lines in the file ABC.A to the beginning of the file, then
copy them into a new file WZ.A.

• Move the text to the beginning of the file:

Move the cursor to the "T" of TOOLS.	
SELECT BLOCK	Activate the selecting function.
↓ repeatedly	Move the cursor to the end of the block.
REMOVE BLOCK	Erase the text and store temporarily.
↑ repeatedly	Move the cursor to the beginning of the file.
INSERT BLOCK	Insert the stored text block. Note: The stored block is inserted above the cursor and may be off screen.
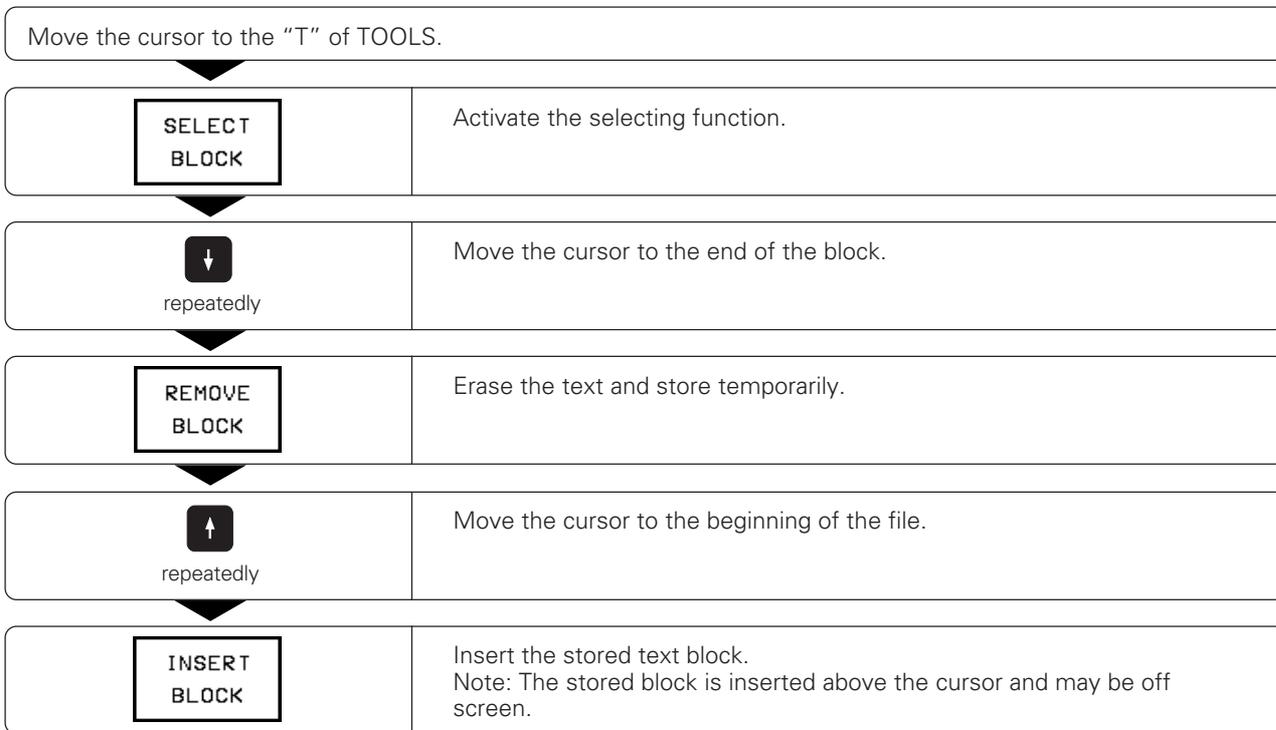
• Select the text again and copy it into another file:

Mark the text block as described above.	
APPEND TO FILE	Select the function for copying to another file.
DESTINATION FILE =	
W Z	Write the name of the file into which you wish to copy the block, for example WZ.
ENT	Copy into a another file. Text block remains marked.
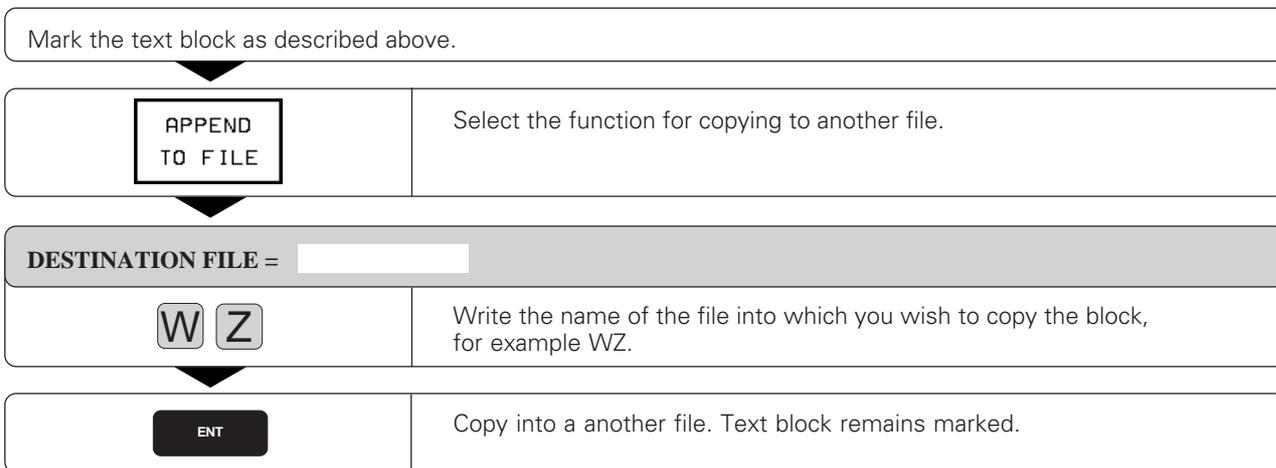
4.11 Creating Pallet Files

Pallet files are used with machining centers, and contain the following information:

- Pallet number   PAL
- Part program name   PGM NAME
- Datum table   DATUM

To edit pallet files:

PROGRAMMING AND EDITING	
PGM NAME	Call the file management.

SELECT TYPE + SHOW .P	Shift the soft-key row and show .P type pallet files.

FILE NAME = _____ .P

Select a pallet file, or enter a new file name to create a new file.
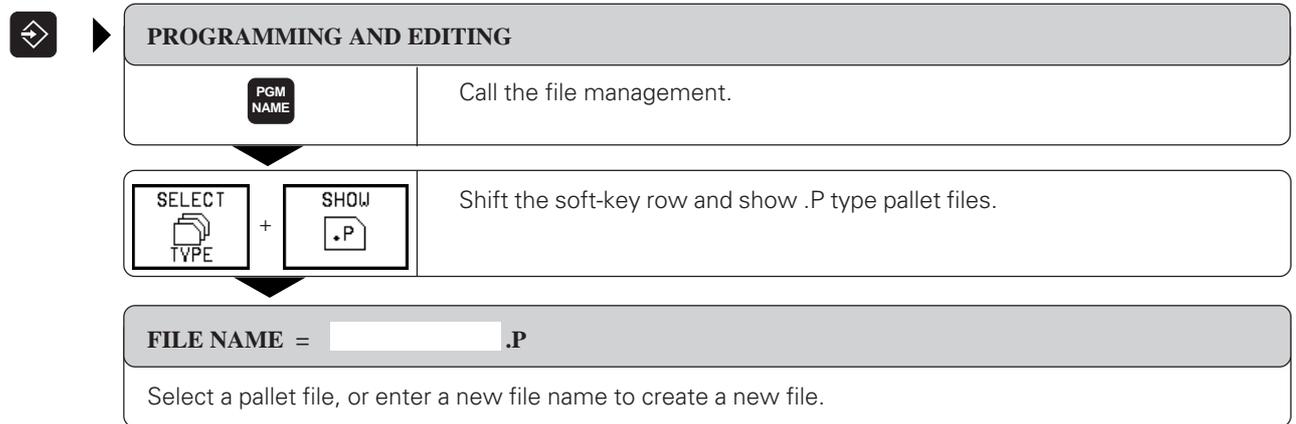
To link programs and datum tables:

PROGRAM NAME ?

Enter the name of a part program that belongs to this pallet file.

DATUM TABLE ?

Enter the name of the datum table for the program.

if necessary	
NEXT LINE	Create more pallet files.
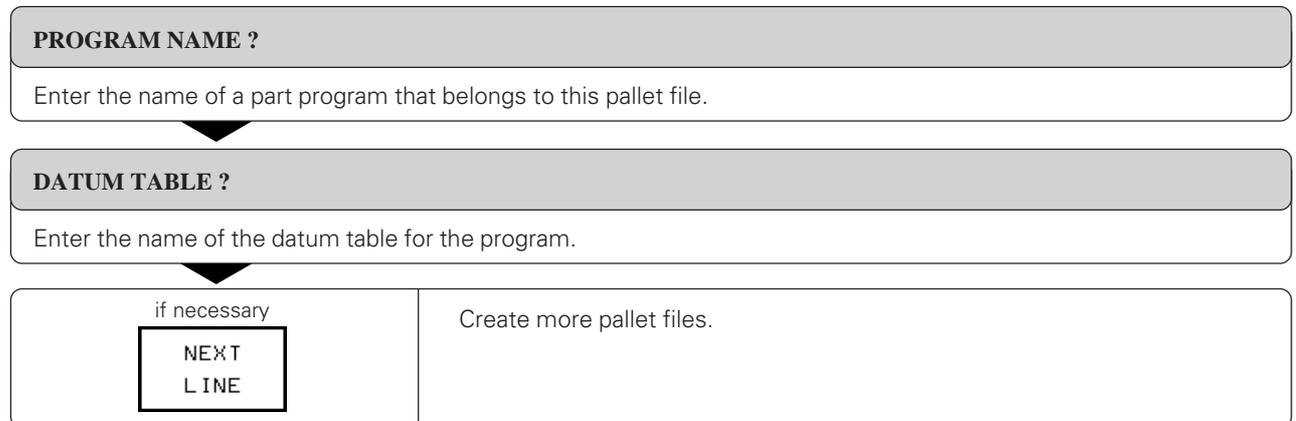
Pallet files are managed and output as determined in the PLC. The machine manual provides further information on this.

Functions for editing pallet tables

The following functions help you to create and change pallet tables:
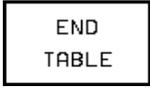
Function	Key/ Soft key
Move the highlight vertically	↑ ↓
Move the highlight horizontally	← →
Go to the beginning of the table	BEGIN TABLE
Go to the end of the table	END TABLE
Go to the next page of the table	PAGE ⇩
Go to the previous page of the table	PAGE ⇧
Insert the last line in the table	INSERT LINE
Delete the last line in the table	DELETE LINE
Go to the beginning of the next line	NEXT LINE
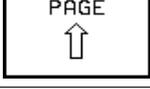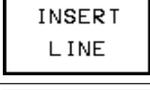
To leave the pallet file:

PGM NAME ▶ SELECT TYPE	SHOW .I	Select a different type of file, for example programs in ISO format.
PGM MGT		
↑ ↓ + SELECT		Choose the desired program.
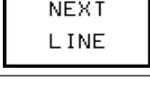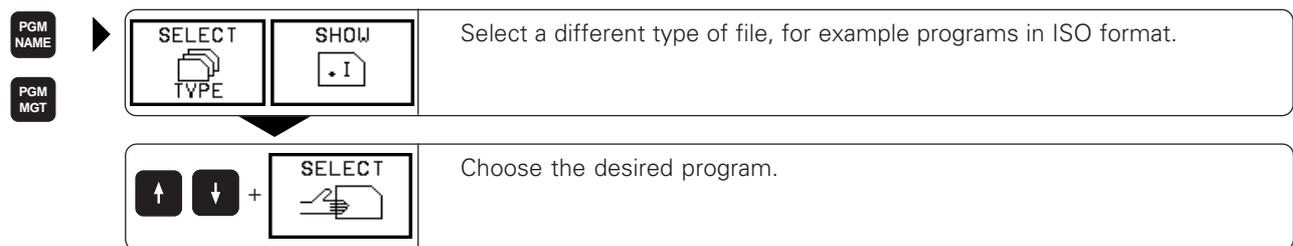
4.12 Adding Comments to the Program

Comments can be added to the part program in the PROGRAMMING AND EDITING mode of operation.

Applications:

- Explanations of program steps
- Adding general notes

Adding comments to program blocks

You can add comments to a program block immediately after entering the data by pressing the semicolon key (;) on the alphabetic keyboard.

Input:

- Enter your comment and conclude the block by pressing the END key.

To add a comment to a block that has already been entered, select the block and press a horizontal arrow key until the semicolon and the dialog prompt appear.

```
MANUAL       PROGRAMMING AND EDITING
OPERATION
N20 G01 Z-25 F250 *
N20 G83 P01 -1 P02 -25 P03 -5 P04 0
    P05 150 *
N20 X+10 Y+52 M3 *
N20 M99 *
N20 X+25 Y+90 M99 *
N20 G98 L1 *
N20 G91 X+10 M99 *
N20 L1,7 *
N20 G90 X+100 Y+50 *
N20 G00 Z+100 *
N20 X+150 Y+25,5 *
N20 G84 P01 -2 P02 -20 P03 0 P04 250 *
N20 Z+2 M3 *
N20 M99 *

PARA-
METER
```

Fig. 4.19:   Dialog for entering comments

To enter a comment as a separate block:

;	Start a new block by pressing the semicolon key.

Enter your comment with the alphabetic and numeric keypads.

END	Close the block.
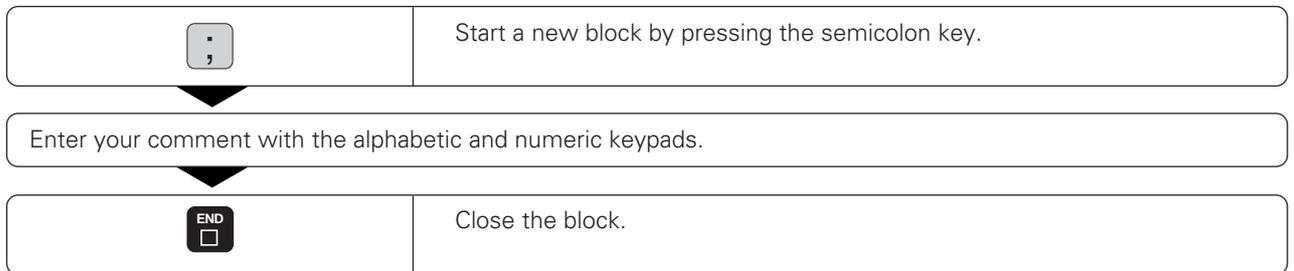
☞ Comments are added behind the entered blocks.

Example

```
    .
    .
    .
N50     G00  X+0 Y–10 ✳
;       PRE-POSITIONING.................... A comment is indicated by a semicolon at the beginning of the block.
N60     G01 G41 F100 ✳
    .
    .
    .
```

5.1 General Information on Programming Tool Movements

Tool movements are always programmed as if the tool moves and the workpiece remains stationary.

☞ Before running a part program, always pre-position the tool to prevent the possibility of damaging it or the workpiece. Radius compensation and a path function must remain active.

*Example NC block: N30 G00 G40 G90 Z+100 ***

Path functions

Each element of the workpiece contour is entered separately using path functions.
You enter:

• Straight lines
• Circular arcs

You can also program a combination of the two contour elements (helical paths).
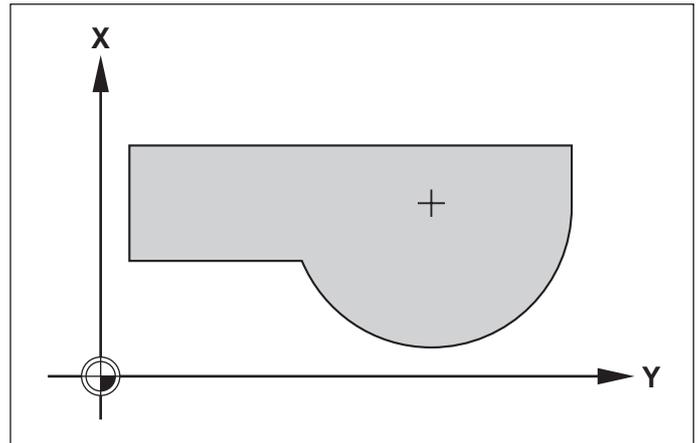
Fig. 5.1: A contour consists of straight lines and circular arcs

The contour elements are executed in sequence to machine the programmed contour.
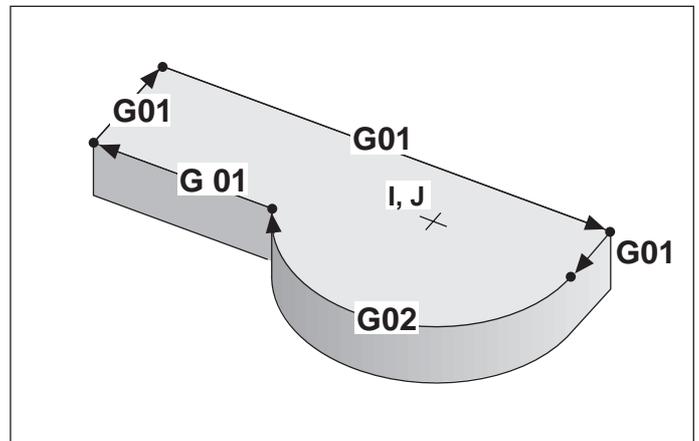
Fig. 5.2: Contour elements are programmed and executed in sequence

Subprograms and program section repeats

If a machining sequence occurs several times in a program, you can save time and reduce the chance of programming errors by entering the sequence once and then defining it as a subprogram or program section repeat.

Programming variants:

- Repeating a machining routine immediately after it is executed (program section repeat)
- Inserting a machining routine at certain locations in a program (subprogram)
- Calling a separate program for execution or test run within the main program (program call)

Cycles

Common machining routines are delivered with the control as standard cycles for:

- Peck drilling
- Tapping
- Slot milling
- Pocket and island milling

Coordinate transformation cycles can be used to change the coordinates of a machining sequence in a defined way. Examples:

- Datum shift
- Mirroring
- Basic rotation
- Enlarging and reducing

Parametric programming

Instead of programming numerical values, you enter markers called *parameters* which are defined through mathematical functions or logical comparisons. You can use parametric programming for:

- Conditional and unconditional jumps
- Measurements with the 3D touch probe during program run
- Output of values and messages
- Transferring values to and from memory

The following mathematical functions are available:

- Assign
- Addition/Subtraction
- Multiplication/Division
- Angle measurement/Trigonometry

among others.

5.2 Contour Approach and Departure

A convenient way to approach or depart the workpiece is on an arc which is tangential to the contour. This is carried out with the approach/departure function G26 (see page 5-6).

Starting point and end point

Starting point

From the starting point, the tool moves to the first contour point. The starting point is programmed without radius compensation.

The starting point must be:

• Approachable without collision
• Near the first contour point
• Located in relation to the workpiece such that no contour damage occurs when the contour is approached.

If the starting point is located within the shaded area of fig. 5.4, the contour will be damaged when the first contour point is approached. The optimum starting point Ⓢ is located in the extension of the tool path for machining the first contour.
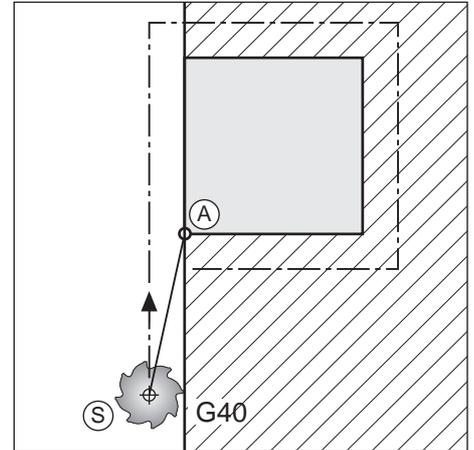
Fig. 5.3 :   Starting point Ⓢ of machining

First contour point

Machining begins at the first contour point. The tool moves to this point with radius compensation.
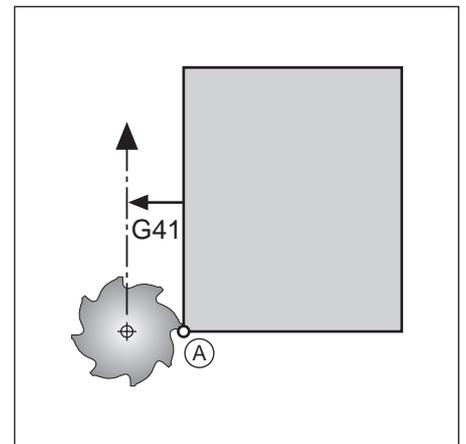
Fig. 5.4 :   First contour point for machining

Approaching the starting point in the spindle axis

When the starting point Ⓢ is approached, the spindle axis is moved to working depth.

If there is danger of collision, approach the starting point in the spindle axis separately.

Example: G00 G40   X ... Y ...   Positioning X/Y
                        Z–10   Positioning Z
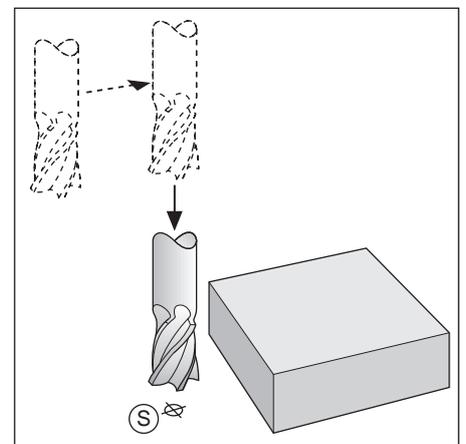
Fig. 5.5 :   Separate movement of the spindle when there is danger of collision

End point

Similar requirements hold for the end point:

• Can be approached without collision
• Near the last contour point
• Avoids tool damage

The ideal location for the end point Ⓔ is again in the extension of the tool path outside of the shaded area. It is approached without radius compensation.
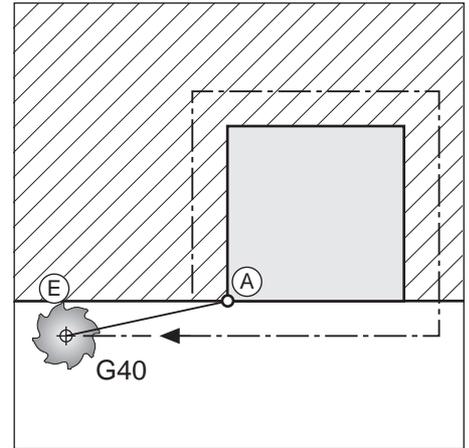
Fig. 5.6 :   End point Ⓔ for machining

Departure from an end point in the spindle axis

The spindle axis is moved separately.

Example: G00 G40   X ... Y ...   Approach end point
                          Z+50   Retract tool
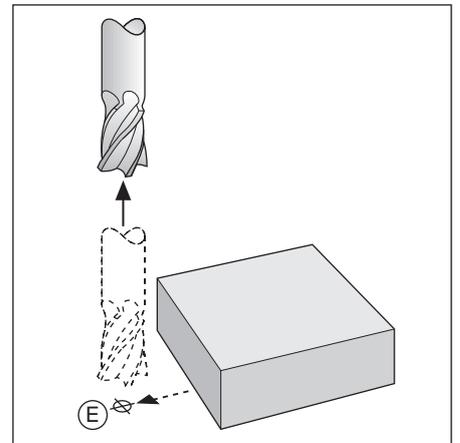
Fig. 5.7 :   Retract spindle axis separately

Common starting and end point

Outside of the shaded areas in the illustrations, it is possible to define a single point as both the starting and end point ⓈⒺ.

The ideal location for the starting and end point is exactly between the extensions of the tool paths for machining the first and last contour elements.

A common starting and end point is approached without radius compensation.
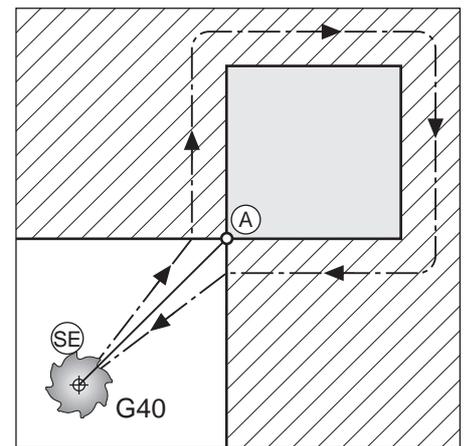
Fig. 5.8:   Common starting and end point

Tangential approach and departure

The tool approaches the contour on a tangential arc with G26, and departs it with G27. This prevents dwell marks.

Starting point and end point

Starting point Ⓢ and end point Ⓔ of the machining sequence are off the workpiece near the first or last contour element.

The tool path to the starting point or end point is programmed without radius compensation.

Input

- For the approach path, G26 is programmed after the block containing the first contour point (the first block with radius compensation G41/G42).
- For the departure path, G27 is programmed after the block containing the last contour point (the last block with radius compensation G41/G42).
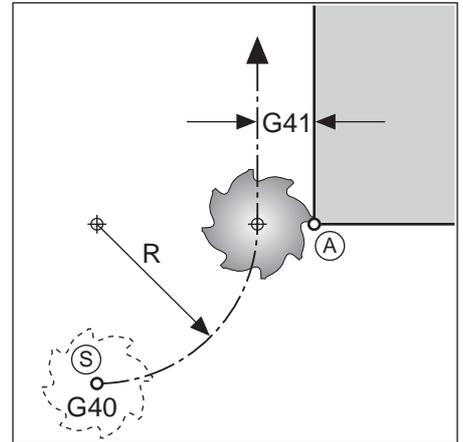
Fig. 5.9:   Soft contour approach
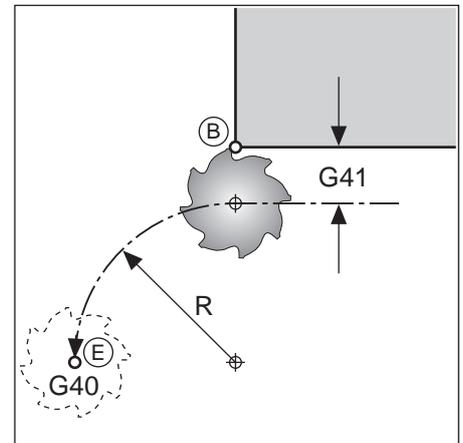
Fig. 5.10:   Soft contour departure

Program structure

```
.
.
.
G00 G40 G90 X ... Y ... ............................................Starting point Ⓢ
G01 G41 X ... Y ... F350 ............................................First contour point Ⓐ
G26 R ... ....................................................................Soft approach
.
.
.
Contour elements
.
.
.
X ... Y ... ...................................................................Last contour point Ⓑ
G27 R ... ....................................................................Soft departure
G00 G40 X ... Y ... ....................................................End point Ⓔ
```

The radius in G26/G27 must be selected such that it is possible to perform the circular arc between the contour point and the starting point or end point.

5.3 Path Functions

General information

Part program input

You create a part program by entering the workpiece dimensions. Coordinates are programmed as absolute values (G90) or relative values (G91).

In general, you program the coordinates of the end point of the contour element.

The TNC automatically calculates the path of the tool based on the tool data and the radius compensation.

Machine axis movement under program control

All axes programmed in a single block are moved simultaneously.

Paraxial movement

The tool moves in a path parallel to the programmed axis.

Number of axes programmed in the block: 1
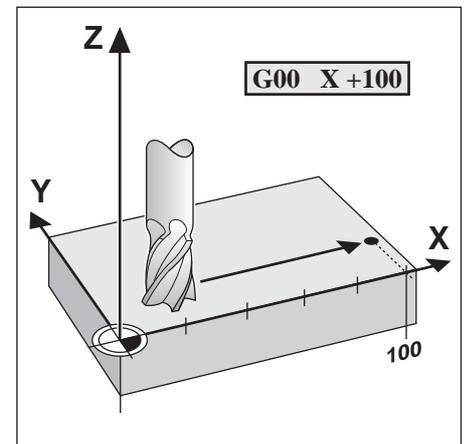
Fig. 5.11:   Paraxial movement

Movement in the main planes

The tool moves to the programmed position on a straight line or circular arc in a plane.

Number of axes programmed in the block: 2
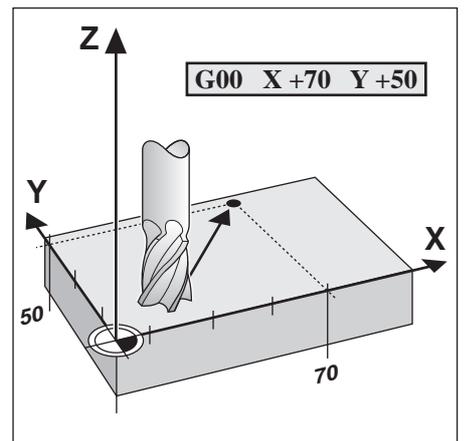
Fig. 5.12:   Movement in a main plane (XY)

Movement of three machine axes (3D movement)

The tool moves in a straight line to the programmed position.

Number of axes programmed in the block: 3

Exception: A helical path is created by combining a circular with a linear
              movement.
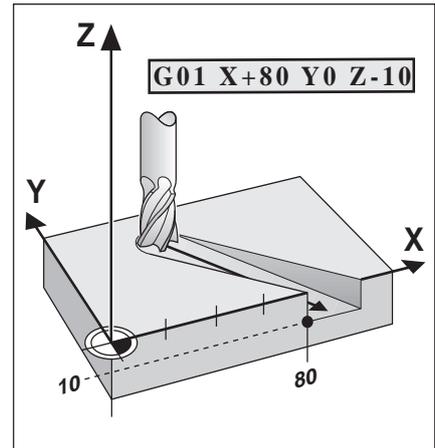
Fig. 5.13:   Three-dimensional movement

Entering more than three coordinates

The TNC can control up to five axes simultaneously
(for example, three linear and two rotary axes).

Such programs are too complex to program at the
machine, however.

Advantages of five-axis machining of 3D surfaces:

• Cylindrical end mills can be used (inclined-tool
  milling)
• Faster machining
• Better surface definition
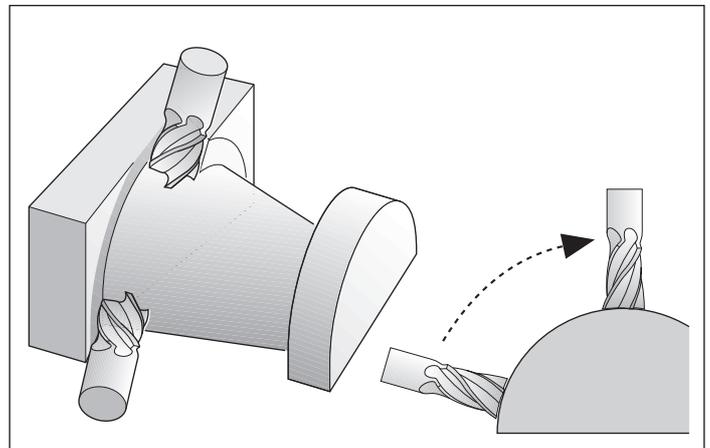
Fig. 5.14:   Example of simultaneous movement of more than three
              axes: machining a 3D surface with an end mill
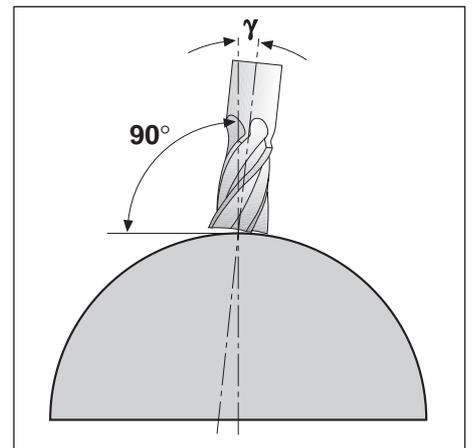
Fig. 5.15:   Inclined-tool machining

Input example:

G01 G40 X+20 Y+10 Z+2 A+15 C+6 F100 M3
(three linear and two rotary axes)

The additional coordinates are programmed as usual in a G01 block.

The TNC graphics cannot simulate four- or five-axis movements.

Overview of path functions

Function	Input in Cartesian coordinates		Input in polar coordinates
Straight line at rapid traverse Straight line at programmed feed rate	G00 G01		G10 G11
Chamfer with length R. A chamfer is inserted between two straight lines.		G24	
Circle center – also the pole for polar coordinates. I,J,K generates no movement.	I, J, K		
Circular arc, clockwise (CW) Circular arc, counterclockwise (CCW) Programming of the circular path: • Circle center I, J, K and end point, or • Circle radius and end point.	G02 G03		G12 G13
Circular movement without direction of rotation. The circular path is programmed with the radius and end point. The direction of rotation results from the last programmed circular movement G02/G12 or G03/G13.	G05		G15
Circular movement with tangential connection. An arc with tangential transition is inserted into the preceding contour element. Only the end point of the arc has to be programmed.	G06		G16
Corner rounding with radius R. An arc with tangential transitions is inserted between two contour elements.		G25	

5.4 Path Contours – Cartesian Coordinates

G00: Straight line with rapid traverse

G01: Straight line with feed rate F ...

To program a straight line, you enter:

- The coordinates of the end point ⓔ of the straight line
- If necessary:
  radius compensation, feed rate, miscellaneous function

The tool moves in a straight line from its current position to the end point ⓔ. The starting position ⓢ is approached in the preceding block.
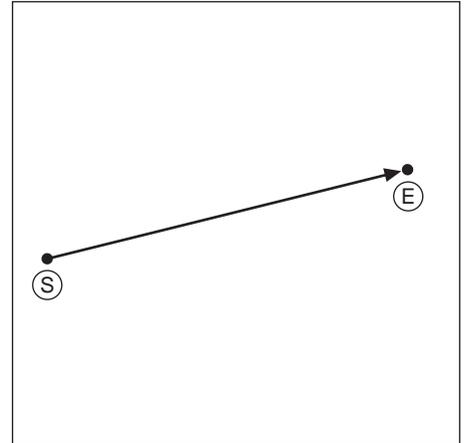
Fig. 5.16:   Linear movement

To program a straight line:

G **0** **0**	Straight line with rapid traverse.

If necessary **G** **9** **1**	Specify as relative coordinate, for example G91 X–50 mm.
X	Select the axis (orange-colored axis key), for example X.
5 **0**	Enter the coordinate of the end point.
If necessary **–/+**	For a negative coordinate, press the +/– key once, e.g. X = –50 mm.
Y . . . **Z**	Enter all further coordinates of the end point.

5.4   Path Contours – Cartesian Coordinates

.
.
.

G 4 1	The TNC moves the tool with radius compensation left of the programmed contour.
G 4 2	The TNC moves the tool with radius compensation right of the programmed contour.
G 4 0	The TNC moves the tool center directly to the end point.
M 3 ENT	Enter miscellaneous function, for example M3 (spindle on, clockwise rotation).
END	When all coordinates have been entered, conclude the block with END.

*Resulting NC block: N25 G00 G42 G91 X+50 G90 Y+10 Z–20 M3 *

Example for exercise: Milling a rectangle

Coordinates of the corner points:

①	X = 5 mm	Y = 5 mm
②	X = 5 mm	Y = 95 mm
③	X = 95 mm	Y = 95 mm
④	X = 95 mm	Y = 5 mm

Milling depth: Z = –10 mm
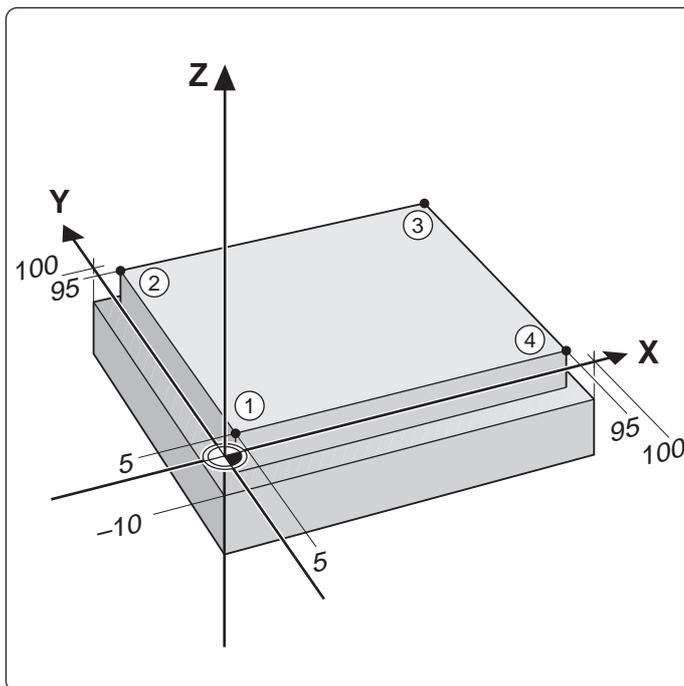
Part program

```
%S512I G71 * .......................................................  Begin the program. Program name S512I,
                                                                      dimensions in millimeters
N10 G30 G17 X+0 Y+0 Z–20 *
N20 G31 G90 X+100 Y+100 Z+0 * ...........................  Define blank form for graphic workpiece simulation
                                                                      (MIN and MAX point)
N30 G99 T1 L+0 R+5 * ............................................  Define tool in the program
N40 T1 G17 S2500 * ...............................................  Call tool in the infeed axis Z (G17);
                                                                      Spindle speed S = 2500 rpm
N50 G00 G40 G90 Z+100 M06 * .............................  Retract in the infeed axis; rapid traverse; miscellaneous
                                                                      function for tool change
N60 X–10 Y–10 * .....................................................  Pre-position near the first contour point
N70 Z–10 M03 * ......................................................  Pre-position in the infeed axis, spindle ON
N80 G01 G41 X+5 Y+5 F150 * .................................  Move to ① with radius compensation
N90 Y+95 * ..............................................................  Move to corner point ②
N100 X+95 * ............................................................  Move to corner point ③
N110 Y+5 * ..............................................................  Move to corner point ④
N120 X+5 * ..............................................................  Move to corner point ①, end of machining
N130 G00 G40 X–10 Y–10 M05 * ...........................  Depart the contour, cancel radius compensation, spindle STOP
N140 Z+100 M02 * ..................................................  Retract in the infeed axis, spindle OFF, coolant OFF,
                                                                      program stop, return to block 1
N99999 %S512I G71 * ............................................  End of program
```

G24: Chamfer

The chamfer function enables you to cut off corners at the intersection of two straight lines.
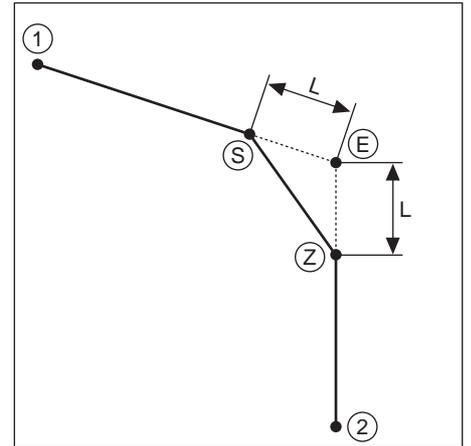
Fig. 5.17:   Chamfer from Ⓢ to Ⓩ

Enter the length *(L)* to be removed from each side of the corner.

Prerequisites

- The radius compensation before and after the chamfer block must be the same
- An inside chamfer must be large enough to accommodate the current tool.
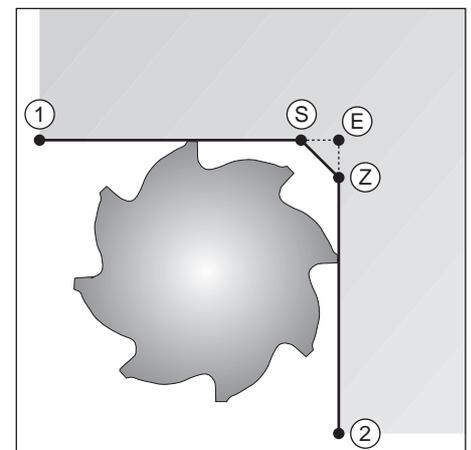
Fig. 5.18:   Tool radius too large

- You cannot start a contour with a G24 block.
- A chamfer is only possible in the working plane.
- The feed rate for chamfering is the same as in the previous block.
- The corner point E is cut off by the chamfer and is not part of the contour.

To program a chamfer:

G 2 4 ENT	Select the chamfer function.

CHAMFER SIDE LENGTH ?	
5 END	Enter the length to be removed from each side of the corner, for example 5 mm.

Resulting NC block: G24 R5*

Example for exercise: Chamfering a corner

Coordinates of the
corner point: (E)   X =   95 mm
                               Y =   5 mm

Length of chamfer: CHF =   10 mm

Tool radius: R = +10 mm

Milling depth: Z = –15 mm

Part program

%S514I G71 *	Begin the program
N10 G30 G17 X+0 Y+0 Z–20 *	Workpiece blank MIN point
N20 G31 G90 X+100 Y+100 Z+0 *	Workpiece blank MAX point
N30 G99 T5 L+5 R+10 *	Define the tool
N40 T5 G17 S2000 *	Call the tool
N50 G00 G40 G90 Z+100 M06 *	Retract and insert tool
N60 X–10 Y–5 *	Pre-position in the working plane
N70 Z–15 M03 *	Move tool to working depth, move spindle to
N80 G01 G42 X+5 Y+5 F200 *	contour with radius compensation at machining feed rate
N90 X+95 *	First straight line for corner E
N100 G24 R10 *	Insert chamfer with length 10 mm
N110 Y+100 *	Second straight line for corner E
N120 G00 G40 X+110 Y+110 *	Depart the contour, cancel radius compensation
N130 Z+100 M02 *	Retract in the infeed axis
N99999 %S514I G71 *	

Circles and circular arcs

Here the TNC moves two axes simultaneously in a circular path relative to the workpiece.
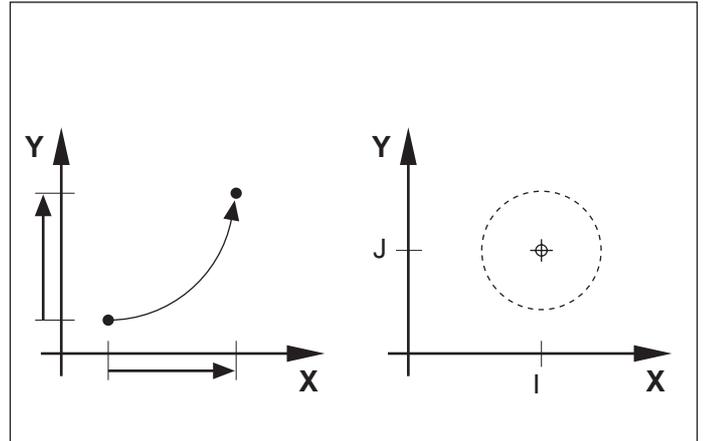
Fig. 5.19:   Circular arc and circle center

Circle center I, J, K

You can define the circle center for circular movement.

A circle center also serves as reference (pole) for polar coordinates.
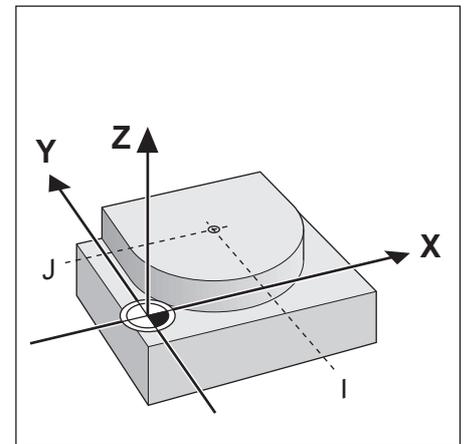
Fig. 5.20:   Circle center coordinates

Direction of rotation

When a circular path has no tangential transition to another contour element, enter the mathematical direction of rotation:

• Clockwise direction of rotation is mathematically negative: G02
• Counterclockwise direction of rotation is mathematically positive: G03
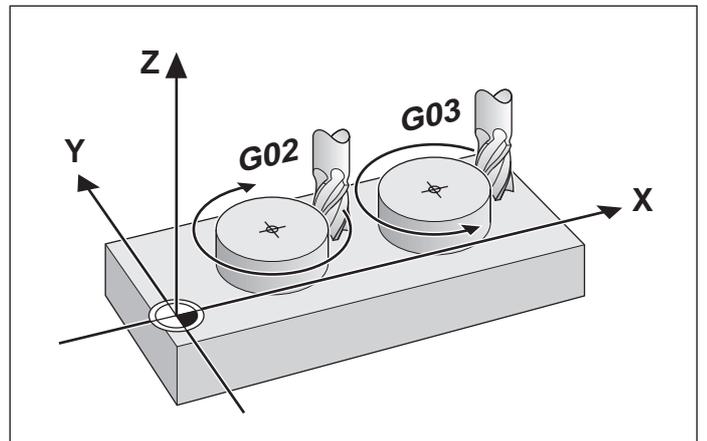
Fig. 5.21:   Direction of rotation for circular movement

Radius compensation in circular paths

You cannot begin radius compensation in a circle block – it must be activated beforehand in a line block.

Circles in the main planes

When you program a circle, the TNC assigns it to one of the main planes. This plane is automatically defined when you set the spindle axis during a tool call (T).

Spindle axis	Main plane	Circle center
Z	XY G17 (UV, XV, UY)	I J
Y	ZX G18 (WU, ZU, WX)	K I
X	YZ G19 (VW, YW, VZ)	J K

Fig. 5.22:   Defining the spindle axis also defines the main plane and the circle center designation

☞ You can program circles that do not lie parallel to a main plane by using Q parameters (see Chapter 7).

Circle center I, J, K

For arcs programmed with G02/G03/G05, it is necessary to define the circle center. This is done in the following ways:

- Entering the Cartesian coordinates of the circle center
- Using the circle center defined in an earlier block
- Capturing the actual position

If G29 is programmed, the last programmed position is automatically used as the circle center or pole.
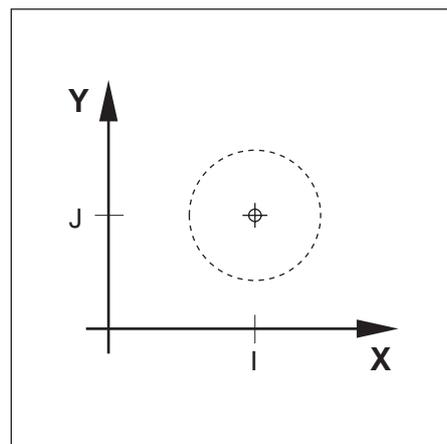
Fig. 5.23:   Circle center I, J

Duration of circle center definition

A circle center definition remains in effect until a new circle center is defined.

5.4   Path Contours – Cartesian Coordinates

Entering I, J, K incrementally

If you enter the circle center with incremental coordinates, you have programmed it relative to the last programmed position of the tool.
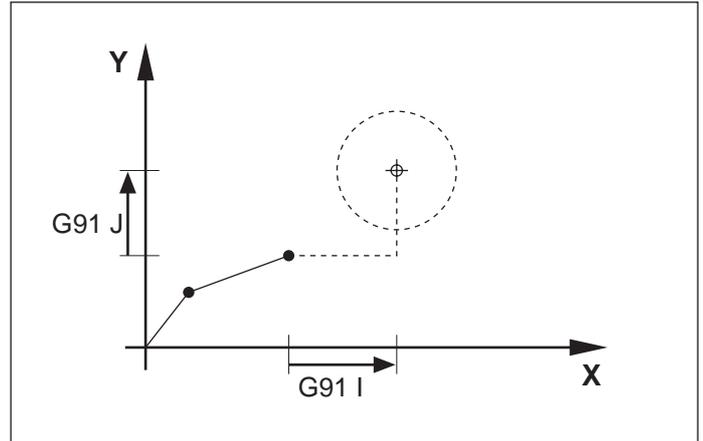
Fig. 5.24:   Incremental coordinates for a circle center

☞
- The circle center I, J, K also serves as the pole for polar coordinates.
- The only effect of I, J, K is to define a position as a circle center — the tool does *not* move to the position.

To program a circle center (pole):

I **2** **0**	Select the first circle center designation, for example I. Enter the coordinate, for example I = 20 mm.
J **1** **0** **⁻/₊** **END ☐**	Select the second circle center designation, for example J. Enter the coordinate, for example J = –10 mm.

Resulting NC block: I+20 J–10 *

5.4   Path Contours – Cartesian Coordinates

G02/G03/G05: Circular path around pole I, J, K

Prerequisites

The circle center I, J, K must be previously defined in the program.
The tool is at the circle starting point Ⓢ.

Defining the direction of rotation

Direction of rotation:

- Clockwise   G02
- Counterclockwise   G03
- No definition   G05
  (the last programmed direction of rotation is used)

Input

- End point of the arc
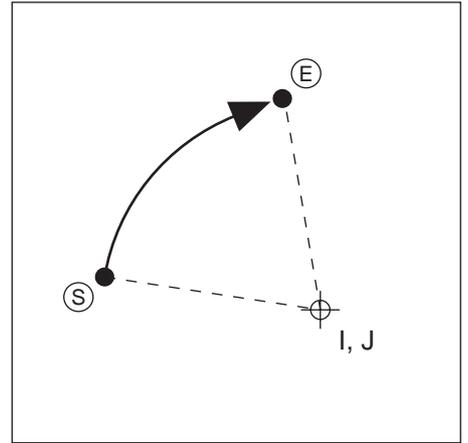
Fig. 5.25:   Circular path from Ⓢ to Ⓔ
              around I, J

The starting and end points of the arc must lie on the circle.
Input tolerance: up to 0.016 mm (selected with MP 7431).

- For a full circle, the end point in the G02/G03 block should be the
  same as the starting point of the contour.
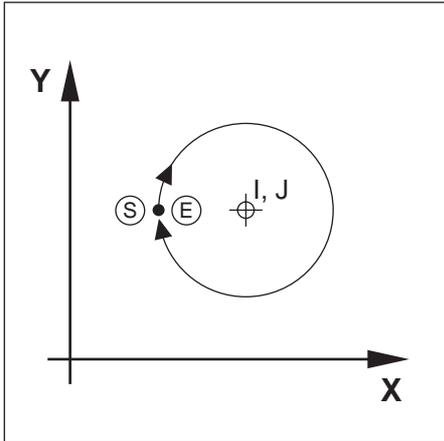
Fig. 5.26:   Full circle around I, J with a G02
              block
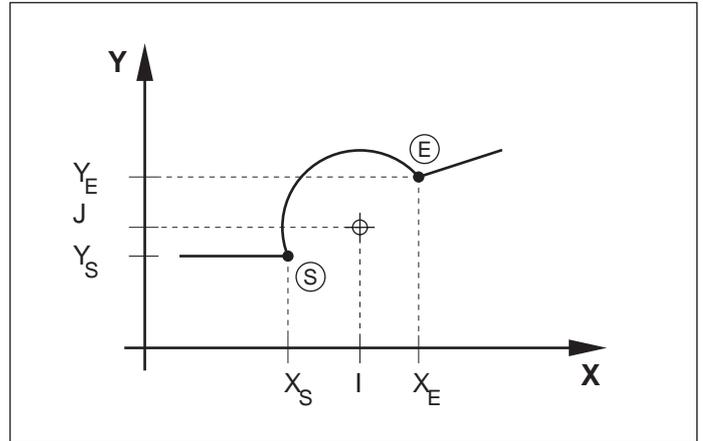
Fig. 5.27:   Coordinates of an arc

To program a circular arc with G02 around a circle center I, J (direction of rotation = clockwise):

G 0 2	Circle in Cartesian coordinates, clockwise.
G 9 1 **X 5**	Enter the first coordinate of the end point in incremental dimensions, for example, X = 5 mm.
G 9 0 **Y 5 ⁻/+** **END**	Enter the second coordinate of the end point in absolute dimensions, for example, Y = –5 mm. Conclude the block.

Further entries, if necessary:

• Radius compensation
• Feed rate
• Miscellaneous function

Resulting NC block: G02 G91 X+5 G90 Y–5

Example for exercise: Mill a full circle with one block

Circle center:	I	= 50 mm
	J	= 50 mm
Beginning and end of the arc:	X	= 50 mm
	Y	= 0 mm
Milling depth:	Z	= –5 mm
Tool radius:	R	= 15 mm
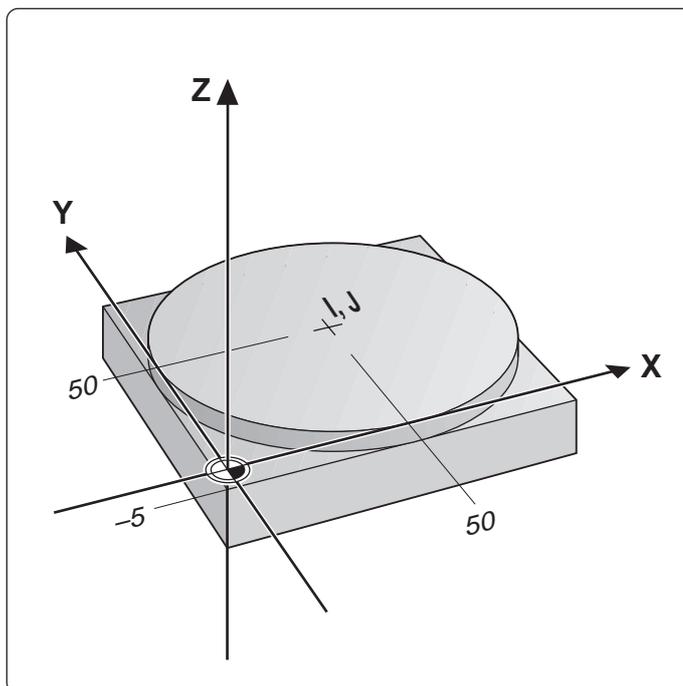
Part program

```
%S520I G71 * ...................................................... Begin the program
N10 G30 G17 X+1 Y+1 Z–20 * ................................. Workpiece blank MIN point
N20 G31 G90 X+100 Y+100 Z+0 * ........................... Workpiece blank MAX point
N30 G99 T6 L+0 R+15 * .......................................... Define the tool
N40 T6 G17 S1500 * ................................................. Call the tool
N50 G00 G40 G90 Z+100 M06 * .............................. Retract and insert tool
N60 X+50 Y–40 * ...................................................... Pre-position in the working plane
N70 Z-5 M03 * .......................................................... Move tool to working depth
N80 I+50 J+50 * ........................................................ Coordinates of the circle center
N90 G01 G41 X+50 Y+0 F100 * ................................ Approach first contour point with radius compensation at
                                                                                       machining feed rate
N100 G26 R10 * ........................................................ Soft (tangential) approach
N110 G02 X+50 Y+0 * .............................................. Mill arc around circle center I,J; direction of rotation negative
                                                                                       (clockwise); coordinates of end point X = +50 mm, Y = +0
N120 G27 R10 * ........................................................ Soft (tangential) departure
N130 G00 G40 X+50 Y–40 * ..................................... Depart the contour, cancel radius compensation
N140 Z+100 M02 * .................................................... Retract in the infeed axis
N99999 %S520I G71 *
```

G02/G03/G05: Circular path with defined radius

The tool moves on a circular path with radius R.

Defining the direction of rotation

- Clockwise   G02
- Counterclockwise   G03
- No definition   G05
  (the last programmed direction of rotation is used)

Inputs

- Coordinates of the end point of the arc
- Radius R of the arc
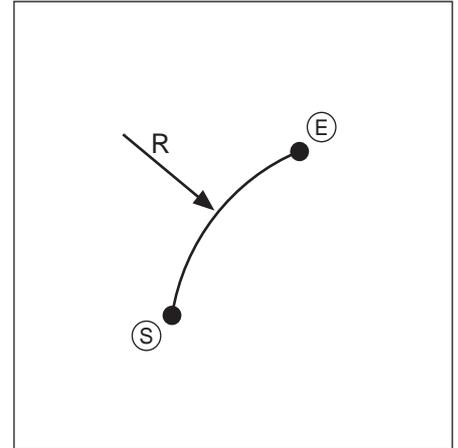
Fig. 5.28:   Circular path from Ⓢ to Ⓔ with radius R

- For a full circle, two G02/G03 blocks must be programmed in succession.
- The distance from the starting and end points of the arc cannot be greater than the diameter of the circle.
- The maximum possible radius is 100 m.
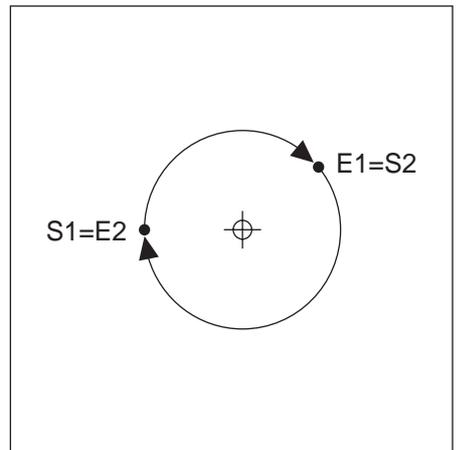
Fig. 5.29:   Full circle with two G02 blocks

Central angle CCA and arc radius R

The starting point Ⓢ and end point Ⓔ on the contour can be connected with four different arcs of the same radius. The arcs have different lengths and curvatures.

Larger arc: CCA>180°
(arc is longer than a semicircle)
Input: Radius R with negative sign (R<0).

Smaller arc: CCA<180°
(arc is shorter than a semicircle)
Input: Radius R with positive sign (R>0).
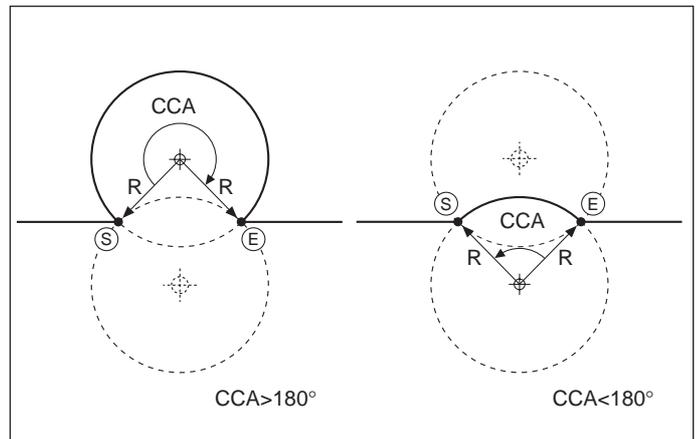
Fig. 5.30:   Arcs with central angles greater than and less than 180°

Contour curvature and direction of rotation

The direction of rotation determines the type of arc:

- Convex (curving outward), or

G02 G41 (R<0)
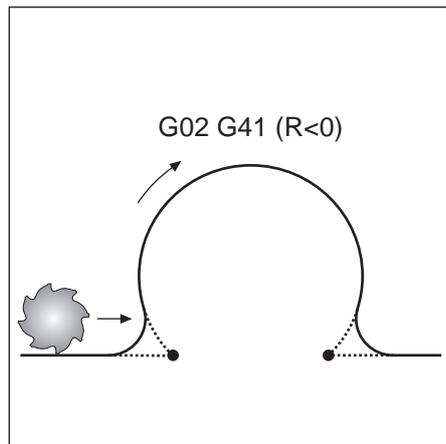
Fig. 5.31:   Convex path

- Concave (curving inward)

G03 G41 (R>0)
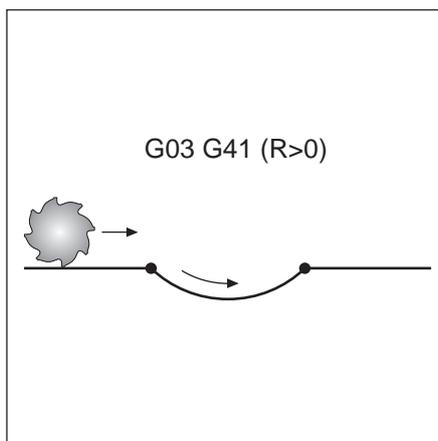
Fig. 5.32:   Concave path

To program a circular arc with a defined radius:

G 0 2	Circle, Cartesian, clockwise.
X 1 0 Y 2	Enter the coordinates of the arc end point, for example X = 10 mm, Y = 2 mm.
R 5 ⁻/₊ ENT	Enter the radius of the arc, for example R = 5 mm, and determine the size of the arc using the sign (negative in this example).

Further entries, if necessary:

- Radius compensation
- Feed rate
- Miscellaneous function

Resulting NC block: G02 G41 X+10 Y+2 R–5

Example for exercise: Milling a concave semicircle

Semicircle radius:	R	= 50 mm
Coordinates of the arc starting point:	X Y	= 0 = 0
Coordinates of the arc end point:	X Y	= 100 mm = 0
Tool radius:	R	= 25 mm
Milling depth:	Z	= –18 mm
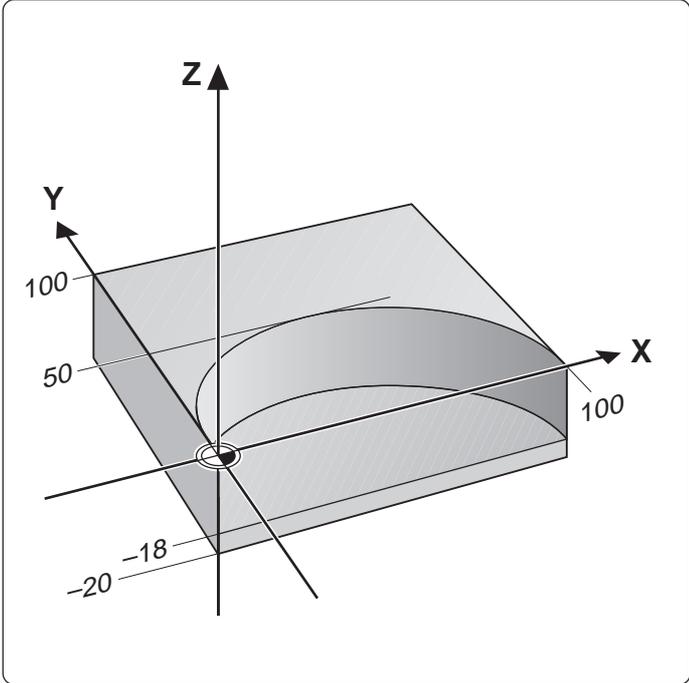
Part program

```
%S523I G71 * ............................................................. Begin the program
N10 G30 G17 X+0 Y+0 Z–20 * .................................. Define the workpiece blank
N20 G31 G90 X+100 Y+100 Z+0 *
N30 G99 T1 L+0 R+25 * ........................................... Define the tool
N40 T1 G17 S780 * .................................................. Call the tool
N50 G00 G40 G90 Z+100 M06 * .............................. Retract and insert tool
N60 X+25 Y–30 * ..................................................... Pre-position in the working plane
N70 Z–18 M03 * ...................................................... Move tool to working depth
N80 G01 G42 X+0 Y+0 F100 * ................................. Approach the contour with radius compensation at
                                                               machining feed rate
N90 G02 X+100 Y+0 R–50 * ..................................... Mill arc to end point X = 100mm, Y = 0;
                                                               radius = 50mm, direction of rotation negative
N100 G00 G40 X+70 Y–30 * ..................................... Depart the contour, cancel radius compensation
N110 Z+100 M02 * ................................................... Retract in the infeed axis
N99999 %S523I G71 *
```

G06: Circular path with tangential connection

The tool moves on an arc that starts at a tangent with the previously programmed contour element.

A transition between two contour elements is tangential when there is no kink or corner at the intersection between the two contours — the transition is smooth.

Input

Coordinates of the end point of the arc.

Prerequisites

- The contour element to which the arc with G06 is to tangentially connect must be programmed directly before the G06 block.
- Before the G06 block there must be at least two positioning blocks defining the contour element which tangentially connects to the arc.
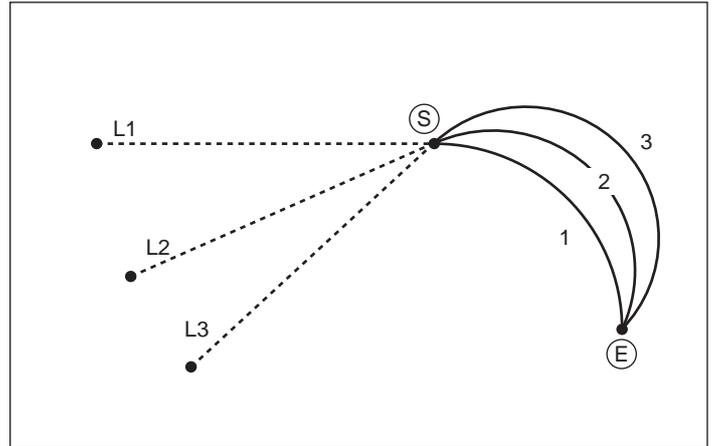
Fig. 5.33:   The straight line ① - ② is connected tangentially to the circular arc Ⓢ - Ⓔ
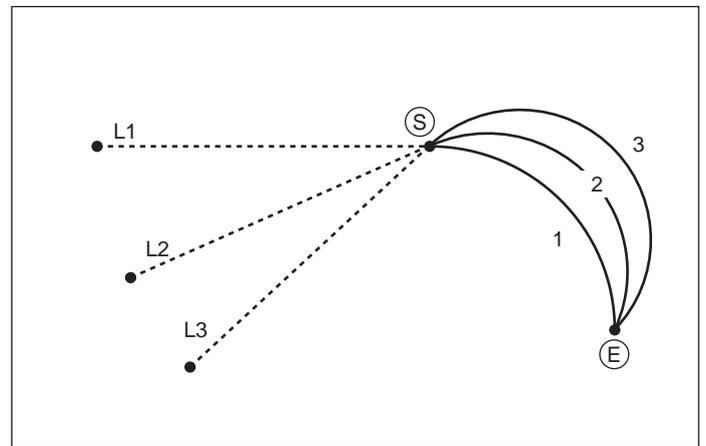
Fig. 5.34:   The path of a tangential arc depends on the preceding contour element

☞ A tangential arc is a two-dimensional operation: the coordinates in the G06 block and in the positioning block preceding it must be in the plane of the arc.

To program a circular path G06 with tangential connection:

G 0 6	Circular path with tangential connection.
G 9 1 X 5 0 Y -/+ 1 0 END	Enter the coordinates of the arc end point in incremental dimensions, for example X = 50 mm, Y = –10 mm.

Further entries, if necessary:

- Radius compensation
- Feed rate
- Miscellaneous function

Resulting NC block: G06 G42 G91 X+50 Y–10 *

Example for exercise: Circular arc connecting to a straight line

Coordinates of the transition
point from the straight
line to the arc: X = 10 mm
                             Y = 40 mm

Coordinates of the
arc end point: X = 50 mm
                             Y = 50 mm

Milling depth: Z = –15 mm

Tool radius: R = 20 mm
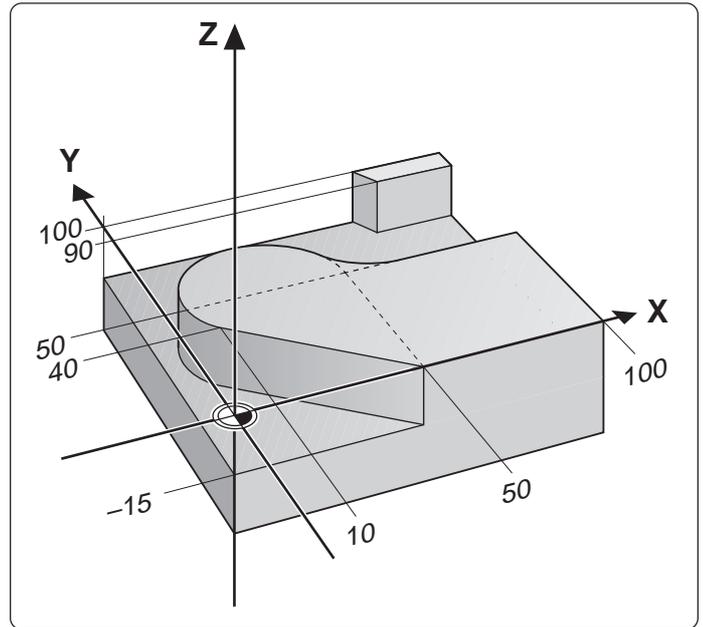
Part program

%S525I G71 *	Begin the program
N10 G30 G17 X+0 Y+0 Z–20 *	Define the workpiece blank
N20 G31 G90 X+100 Y+100 Z+0 *	
N30 G99 T12 L–25 R+20 *	Define the tool
N40 T12 G17 S1000 *	Call the tool
N50 G00 G40 G90 Z+100 M06 *	Retract and insert tool
N60 X+30 Y–30 *	Pre-position in the working plane
N70 Z–15 M03 *	Move the tool to working depth
N80 G01 G41 X+50 Y+0 F100 *	Approach the contour with radius compensation at machining feed rate
N90 X+10 Y+40 *	Straight line to which the arc tangentially connects
N100 G06 X+50 Y+50 *	Arc to end point X = 50 mm, Y = 50 mm; connects tangentially to the straight line in block N90
N110 G01 X+100 *	Complete the contour
N120 G00 G40 X+130 Y+70 *	Depart the contour, cancel radius compensation
N130 Z+100 M02 *	Retract in the infeed axis
N99999 %S525I G71 *	

G25: Corner rounding

The tool moves in an arc that is tangentially connected to both the preceding and following contour elements.

G25 is used to round corners.

Input

• Radius of the arc
• Feed rate for the arc

Prerequisite

The rounding radius must be large enough to accommodate the tool.
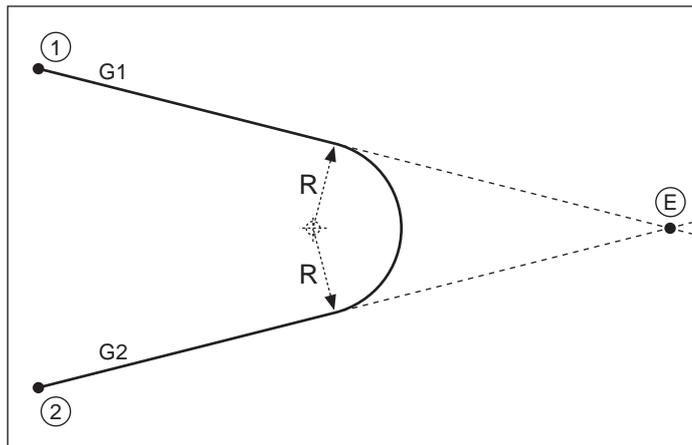
Fig. 5.35:   Rounding radius R between G1 and G2

☞
• In both the preceding and subsequent positioning blocks, both coordinates must lie in the plane of the arc.

• The corner point Ⓔ is not part of the contour.

• A feed rate programmed in a G25 block is effective only in that block. After the G25 block, the previous feed rate becomes effective again.

To program a tangential arc between two contour elements:

G 2 5 ENT	Select the corner-rounding function.

ROUNDING-OFF RADIUS

1 0 ENT	Enter the rounding radius, for example R = 10 mm.

1 0 0 ENT	Enter the feed rate for corner rounding, for example F = 100 mm/min.

Resulting NC block: G25 R 10 F 100

Example for exercise: Rounding a corner

Coordinates of the corner point:	X	=	95 mm
	Y	=	5 mm
Rounding radius:	R	=	20 mm
Milling depth:	Z	=	–15 mm
Tool radius:	R	=	10 mm
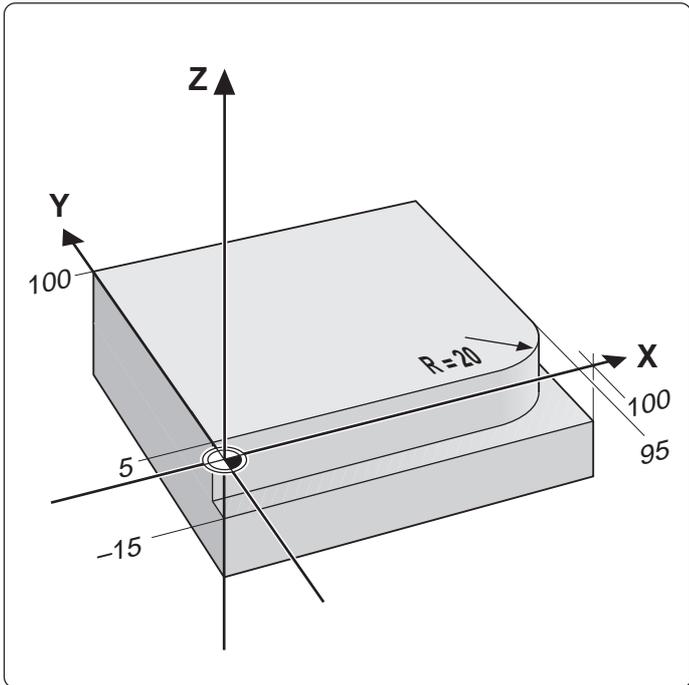
Part program

```
%S527I G71 * ........................................................  Begin the program
N10 G30 G17 X+0 Y+0 Z–20 * ...............................  Define the workpiece blank
N20 G31 G90 X+100 Y+100 Z+0 *
N30 G99 T7 L+0 R+10 * ..........................................  Define the tool
N40 T7 G17 S1500 * .................................................  Call the tool
N50 G00 G40 G90 Z+100 M06 * .............................  Retract and insert tool
N60 X–10 Y–5 * .......................................................  Pre-position in the working plane
N70 Z–15 M03 * ........................................................  Move the tool to working depth
N80 G01 G42 X+0 Y+5 F100 * .................................  Approach the contour with radius compensation at
                                                           machining feed rate
N90 X+95 * ...............................................................  First straight line for the corner
N100 G25 R20 * .......................................................  Insert a tangential arc with radius R = 20 mm between
                                                           the contour elements
N110 Y+100 * ...........................................................  Second straight line for the corner
N120 G00 G40 X+120 Y+120 * ...............................  Depart the contour, cancel radius compensation
N130 Z+100 M02 * ...................................................  Retract in the infeed axis
N99999 %S527I G71 *
```

5.5   Path Contours – Polar Coordinates

Polar coordinates are useful with:

- Positions on circular arcs
- Workpiece drawing dimensions in degrees

Polar coordinates are explained in detail in the section "Fundamentals of NC" (page 1-12).

Polar coordinate origin: Pole I, J, K

The pole can be defined anywhere in the program before blocks containing polar coordinates. Similar to a circle center, the pole is defined in an I, J, K block using its coordinates in the Cartesian coordinate system. The pole remains in effect until a new pole is defined. The designation of the pole depends on the working plane:

Working plane	Pole
XY	I, J
YZ	J, K
ZX	K, J
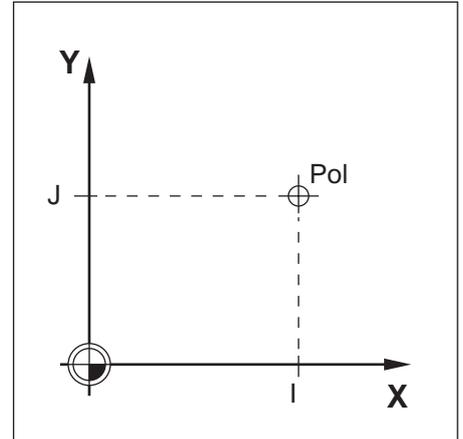
Fig. 5.36:   The pole is the same as a circle center

G10: Straight line with rapid traverse

G11: Straight line with feed rate F …

- Values from –360° to +360° are permissible for the angle H
- The sign of H depends on the angle reference axis:
  Angle from angle reference axis to R is counterclockwise: H>0
  Angle from angle reference axis to R is clockwise: H<0
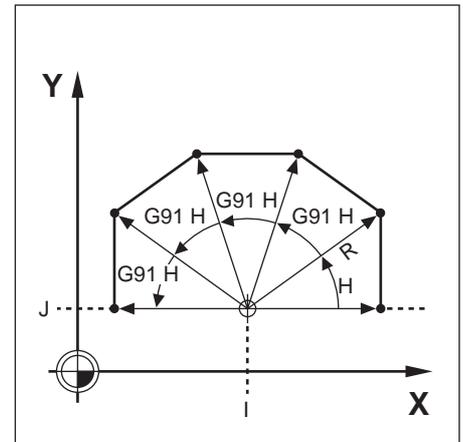
Fig. 5.37:   Contour consisting of straight lines with polar coordinates

G 1 0	Straight line in polar coordinates with rapid traverse.
R 5	Enter radius R from pole to end point of line (here, R = 5 mm).
H 3 0 END	Enter angle H from angle reference axis to R (here, H = 30°).

Resulting NC block: G10 R5 H30 *

Example for exercise: Milling a hexagon

Corner point coordinates:

①	H	= 180°	R	= 45	mm
②	H	= 120°	R	= 45	mm
③	H	= 60°	R	= 45	mm
④	H	= 0°	R	= 45	mm
⑤	H	= 300°	R	= 45	mm
⑥	H	= 240°	R	= 45	mm

Milling depth: Z   = –10 mm

Tool radius: R   =   5 mm
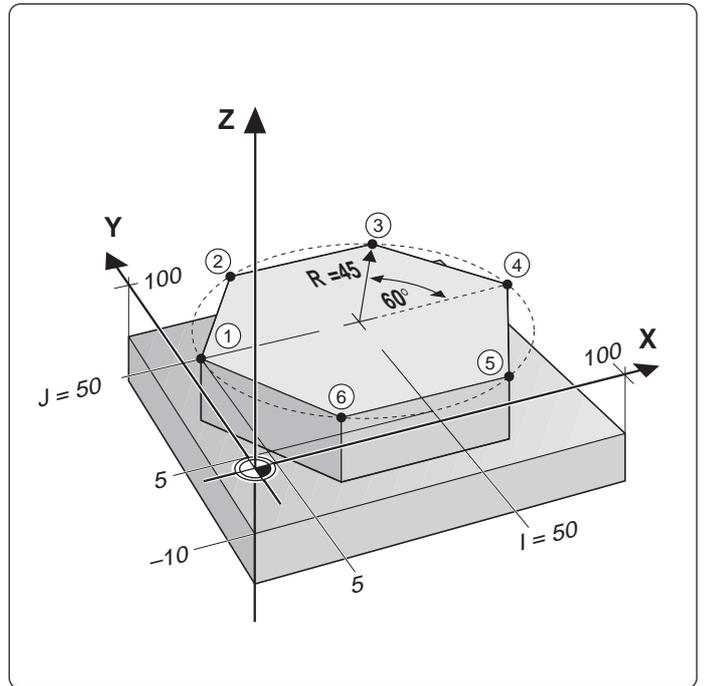
Part program

```
%S530I G71 * ........................................... Begin program
N10 G30 G17 X+0 Y+0 Z–20 * ................................. Define the workpiece blank
N20 G31 G90 X+100 Y+100 Z+0 *
N30 G99 T1 L+0 R+17 * ........................................ Define the tool
N40 T1 G17 S3200 * .................................... Call the tool
N50 G00 G40 G90 Z+100 M06 * ............................. Retract and insert tool
N60 I+50 J+50 * ........................................ Set pole
N70 G10 R+70 H–190 * ............................................ Pre-position in the working plane with polar
                                                               coordinates
N80 Z–10 M03 * ...................................... Move tool to working depth
N90 G11 G41 R+45 H+180 F100 * ........................... Move to contour point 1
N100 H+120 * ........................................... Move to contour point 2
N110 H+60 * ............................................ Move to contour point 3
N120 G91 H–60 * ................................... Move to contour point 4, incremental dimensions
N130 G90 H–60 * .................................. Move to contour point 5, absolute dimensions
N140 H+240 * .......................................... Move to contour point 6
N150 H+180 * .......................................... Move to contour point 1
N160 G10 G40 R+70 H+170 * ................................. Depart contour, cancel radius compensation
N170 Z+100 M02 * ..................................... Retract in the infeed axis
N99999 %S530I G71 *
```

G12/G13/G15: Circular path around pole I, J, K

The polar coordinate radius is also the radius of the arc. It is defined by the distance from the starting point Ⓢ to the pole.

Input

• Polar coordinate angle H for the end point of the arc

☝ Permissible values for H: –5400° to +5400°
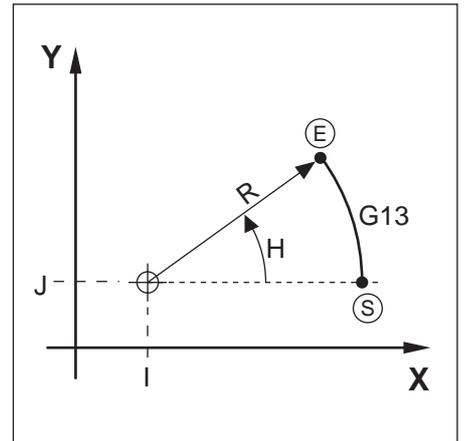
Fig. 5.38: Circular path around a pole

Defining the direction of rotation

Direction of rotation

• Clockwise   G12
• Counterclockwise   G13
• No definition   G15
  (the last programmed direction of rotation is used)

G 1 2	Circle, polar coordinates, clockwise.

H 3 0 END	Enter angle H for the end point of the arc (here, H = 30°). Confirm entry.

Further entries, if necessary:

Radius compensation R
Feed rate F
Miscellaneous function M

*Resulting NC block: G12 H30 ***

Example for exercise: Milling a full circle

Circle center coordinates:	X = 50 mm Y = 50 mm	
Radius:	R = 50 mm	
Milling depth:	Z = – 5 mm	
Tool radius:	R = 15 mm	
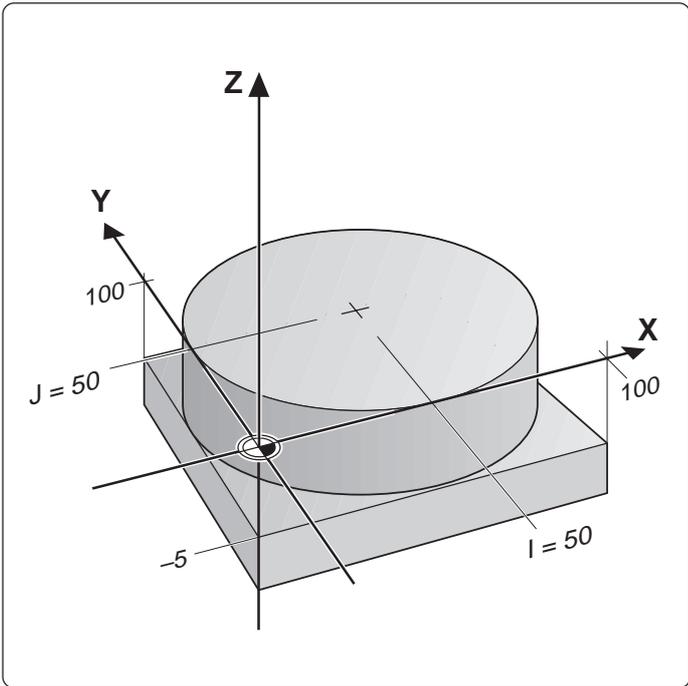
Part program

%S532I G71 *	Begin the program
N10 G30 G17 X+0 Y+0 Z–20 *	Define the workpiece blank
N20 G31 G90 X+100 Y+100 Z+0 *	
N30 G99 T25 L+0 R+15 *	Define the tool
N40 T25 G17 S1500 *	Call the tool
N50 G00 G40 G90 Z+100 M06 *	Retract and insert tool
N60 I+50 J+50 *	Set pole
N70 G10 R+70 H+280 *	Pre-position in the working plane with polar coordinates
N80 Z–5 M03 *	Move tool to working depth
N90 G11 G41 R+50 H–90 F100 *	Approach the contour with radius compensation at machining feed rate
N100 G26 R10 *	Soft (tangential) approach
N110 G12 H+270 *	Circle to end point H = 270°, negative direction of rotation
N120 G27 R10 *	Soft (tangential) departure
N130 G10 G40 R+70 H–110 *	Depart contour, cancel radius compensation
N140 Z+100 M02 *	Retract in the infeed axis
N99999 %S532I G71 *	

G16: Circular path with tangential transition

Moving on a circular path, the tool transitions tangentially to the previous contour element (① to ②) at ②.

Input:

- Polar coordinate angle H of the arc end point Ⓔ
- Polar coordinate radius R of the arc end point Ⓔ
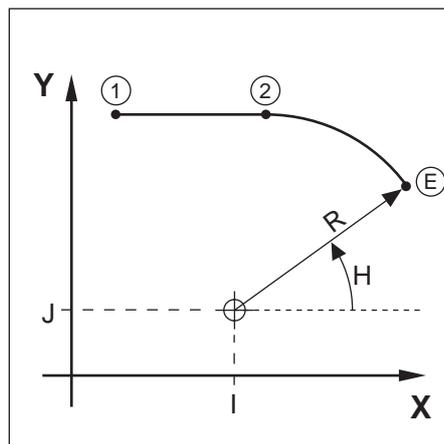
Fig. 5.39:   Circular path around a pole with tangential transition

☞ • The transition point must be exactly defined.
   • The pole is not the center of the contour arc.

G 1 6	Circle, polar coordinates, with tangential transition.
R 1 0	Enter distance R from arc end point to pole (here, R = 10 mm).
H 8 0 END	Enter angle from reference axis to R (here, H = 80°) and confirm entry.

Further entries, if necessary:

Radius compensation R
Feed rate F
Miscellaneous function M

*Resulting NC block: G16 R+10 H+80 ***

Helical interpolation

A helix is a combination of circular motion in a main plane and linear motion in a plane perpendicular to the main plane.

Helices can only be programmed in polar coordinates.

Applications

You can use helical interpolation with form cutters to machine:

- Large-diameter internal and external threads
- Lubrication grooves
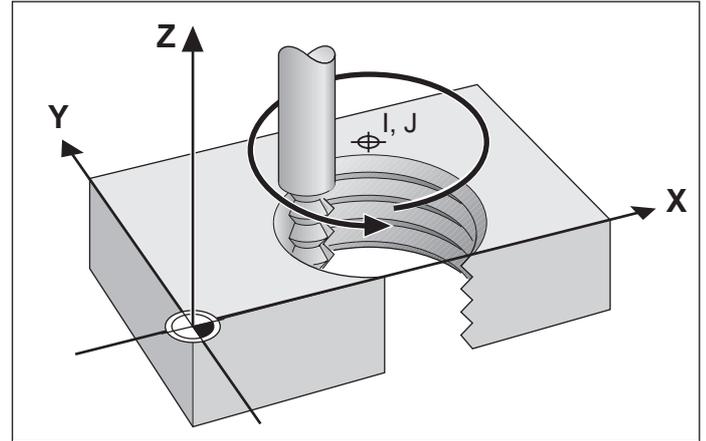
Fig. 5.40: A helix combines circular motion with linear motion

Input

- Total incremental angle of tool traverse on the helix
- Total height of the helix

Total incremental angle

Calculate the total incremental polar angle G91 H as follows:

$$H = n \cdot 360°,$$

where *n* is the number of revolutions of the helical path.

G91 H can be programmed with any value from –5400° to +5400° (i.e., up to *n* = 15).

Total height

Enter the height *h* of the helix referenced to the tool axis. The height is determined as follows:

$$h = n \cdot P,$$

where *n* is the number of thread revolutions and *P* is the thread pitch.

Radius compensation

Enter the radius compensation for the helix according to the table at right.

Internal thread	Work direction	Rotation	Radius comp.
Right-handed	Z+	G13	G41
Left-handed	Z+	G12	G42
Right-handed	Z–	G12	G42
Left-handed	Z–	G13	G41

External thread	Work direction	Rotation	Radius comp.
Right-handed	Z+	G13	G42
Left-handed	Z+	G12	G41
Right-handed	Z–	G12	G41
Left-handed	Z–	G13	G42

Fig. 5.41:   The shape of the helix determines the direction of rotation and the radius compensation

To program a helix:

G 1 2	Helix, clockwise.
G 9 1 H 1 0 8 0	Enter the total angle through which the tool is to move on the helix in incremental dimensions (here, H = 1080°).
Z 4 · 5 END	Enter the height of the helix in the tool axis, likewise in incremental dimensions (here, Z = 4.5 mm). Confirm your entry.

Further entries, if necessary:

Radius compensation
Feed rate F
Miscellaneous function M

*Resulting NC block: G12 G91 H+1080 Z+4.5 ***

Example for exercise: Tapping

Given data

Thread:
Right-handed internal thread M64 x 1.5

Pitch P:	1.5 mm
Starting angle A_S	0°
End angle A_E:	360° = 0° at Z_E = 0
Thread revolutions n_R:	8

Thread overrun:
• at start of thread n_S: 0.5
• at end of thread n_E: 0.5

Number of cuts: 1
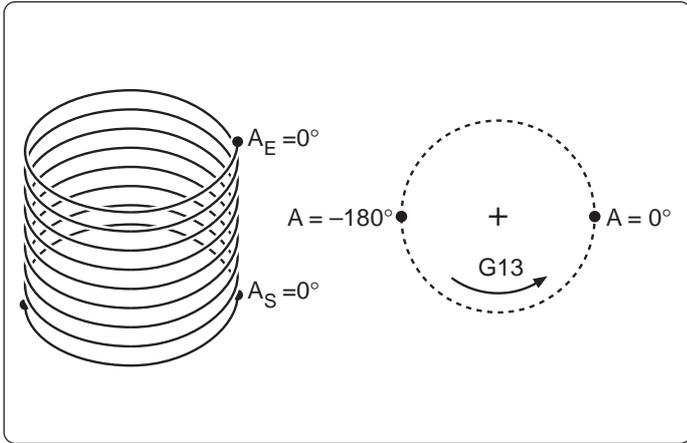
Calculating the input values

• Total height h:

$$h = P \cdot n$$
$$P = 1.5 \text{ mm}$$
$$n = n_R + n_S + n_E = 9$$
$$h = 13.5 \text{ mm}$$

• Incremental polar coordinate angle H:

$$H = n \cdot 360°$$
n = 9 (see total height h)
$$G91\ H = 360° \cdot 9 = 3240°$$

• Starting angle A_S with thread overrun n_S: n_S = 0.5

The starting angle of the helix is advanced by 180° (n = 1 corresponds to 360°). With positive rotation this means
A_S with n_S = A_S – 180° = –180°

• Starting coordinate:

$$Z = P \cdot (n_R + n_S)$$
$$= -1.5 \cdot 8.5 \text{ mm}$$
$$= -12.75 \text{ mm}$$

Z_S is negative because the thread is being cut in an upward direction towards Z_E = 0.

Part program

```
%S536I G71 * ......................................... Begin the program
N10 G30 G17 X+0 Y+0 Z–20 * ................ Define the workpiece blank
N20 G31 G90 X+100 Y+100 Z+0 *
N30 G99 T11 L+0 R+5 * ......................... Define the tool
N40 T11 G17 S2500 * ............................. Call the tool
N50 G00 G40 G90 Z+100 M06 * ............. Retract and insert tool
N60 X+50 Y+30 * ..................................... Pre-position in the working plane to the center of the hole
N70 G29 * ............................................... Transfer position as pole
N80 Z–12 M03 * ..................................... Move tool to starting depth
N90 G11 G41 R+32 H–180 F100 * .......... Approach contour with radius compensation at machining feed rate
N100 G13 G91 H+3240 Z+13.5 F200 *    Helical interpolation; angle and movement in infeed axis are incremental
N110 G00 G40 G90 X+50 Y+30 * ............ Depart contour (absolute), cancel radius compensation
N120 Z+100 M02 * .................................. Retract in the infeed axis
N99999 %S536I G71 *
```

Part program for cutting a thread with more than 15 revolutions (also see Chapter 6)

•
•
```
N80   G00 G40 G90 Z–12.75 M3
N90   G11 G41 R+32 H–180 F100
N100  G26 R+20
N110  G98 L1 ........................................... Identify beginning of program section repeat
N120  G13 G91 H+360 Z+1.5 F200 ......... Enter thread pitch directly as an incremental Z dimension
N130  L 1,24 ........................................... Program the number of repeats (thread revolutions)
N140  G27 R+20
```
•
•

5.6 M Functions for Contouring Behavior and Coordinate Data

The following miscellaneous functions enable you to change the TNC's standard contouring behavior in certain situations:

- Smoothing corners
- Inserting rounding arcs at non-tangential straight-line transitions
- Machining small contour steps
- Machining open contours
- Programming machine-referenced coordinates

Smoothing corners: M90

Standard behavior – without M90

The TNC stops the axes briefly at sharp transitions such as inside corners and contours without radius compensation.
Advantages:

- Reduced wear on the machine
- High definition of corners (outside)

Note:

In program blocks with radius compensation (G41/G42), the TNC automatically inserts a transition arc at outside corners.

Fig. 5.42:   Standard contouring behavior at G40 without M90

Smoothing corners with M90

At corners, the tool moves at constant speed. Advantages:

- A smoother, more continuous surface
- Reduced machining time

Example application:
Surface consisting of a series of straight line segments.

Duration of effect

Servo lag mode must be selected. M90 is only effective in the blocks in which it is programmed.

Independently of M90, you can use machine parameter MP7460 to set a limit value up to which the tool moves at constant path speed (effective with servo lag and feed precontrol). See page 11-14.

Fig. 5.43:   Behavior at G40 with M90

Machining small contour steps: M97

Standard behavior – without M97

The TNC inserts a transition arc at outside corners. If the contour steps are very small, however, the tool would damage the contour. In such cases the TNC interrupts program run and generates the error message TOOL RADIUS TOO LARGE.
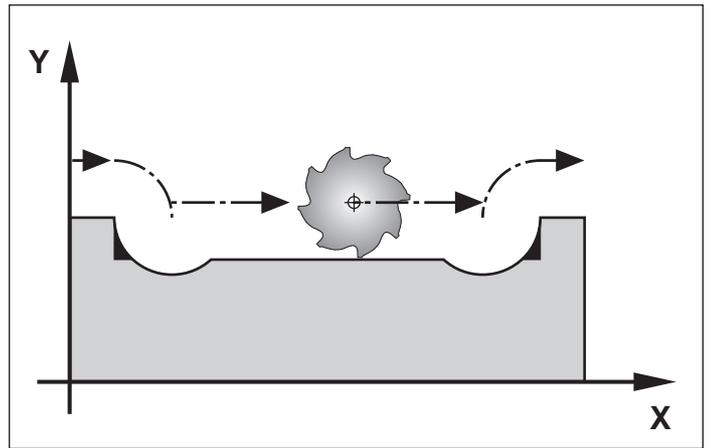
Fig. 5.44:   Standard contouring behavior without M97 when the control would not generate an error message

Machining contour steps – with M97

The TNC calculates the contour intersection Ⓢ (see figure) of the contour elements – as at inside corners – and moves the tool over this point. M97 is programmed in the same block as the outside corner point.

Duration of effect

M97 is effective only in the blocks in which it is programmed.
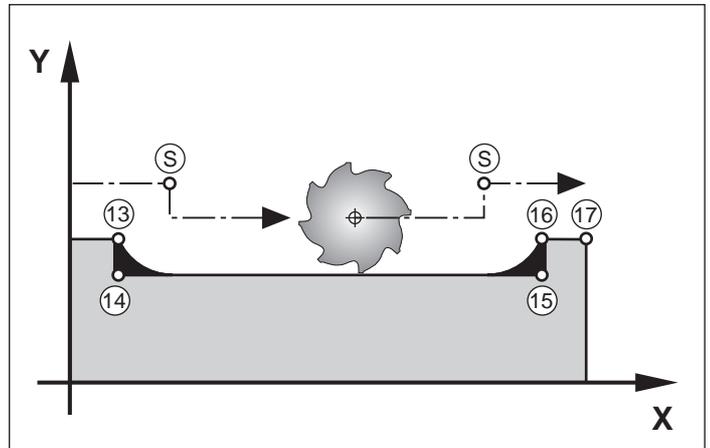
Fig. 5.45:   Contouring behavior with M97

☞ A corner machined with M97 will not be completely finished. It may have to be reworked with a smaller tool.

Program structure

```
.
.
.
N5      G99 L ... R+20 .................................................. Large tool radius
.
.
.
N20     G01 X ... Y ... M97 ........................................... Move to contour point 13
N30     G91  Y–0.5 ...................................................... Machine small contour step 13-14
N40     X+100 ............................................................. Move to contour point 15
N50     Y+0.5 M97 ...................................................... Machine small contour step 15-16
N60     G90 X ... Y ... .................................................. Move to contour point 17
.
.
.
```

The outside corners are programmed in blocks N20 and N50. These are the blocks in which you program M97.

Machining open contours: M98

Standard behavior – without M98

The TNC calculates the intersections ⑤ of the cutter paths and moves the tool in the new direction at those points. If the contour is open at the corners, however, this will result in incomplete machining.
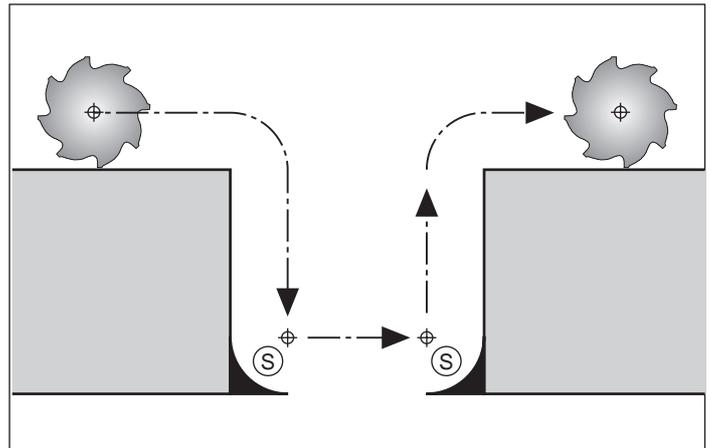
Fig. 5.46:   Tool path without M98

Machining open corners with M98

With M98, the TNC temporarily suspends radius compensation to ensure that both corners are completely machined.

Duration of effect

M98 is effective only in the blocks in which it is programmed.
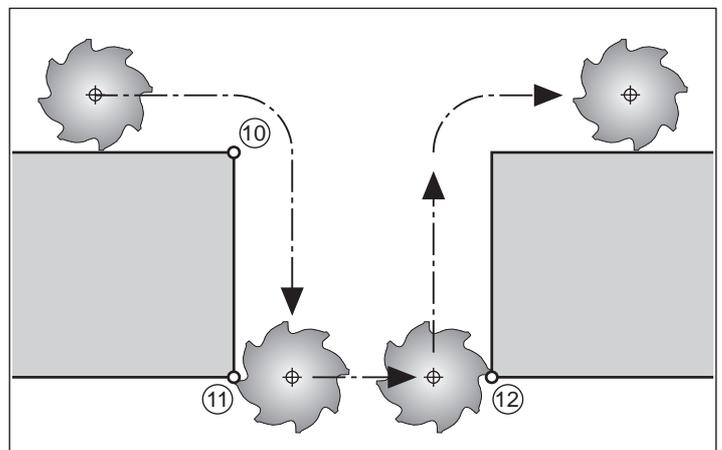
Fig. 5.47:   Tool path with M98

Program structure

```
 .
 .
 .
N10    X ... Y ... G41 F .. ........................................... Move to contour point 10
N20    X ... Y–... M98 ................................................ Machine contour point 11
N30    X + ... ............................................................. Move to contour point 12
 .
 .
 .
```

Programming machine-referenced coordinates: M91/M92

Standard setting

Coordinates are referenced to the workpiece datum (see page 1-13).

Scale reference point

The position feedback scales are provided with one or more reference marks. Reference marks define the position of the scale reference point. If the scale has only one reference mark, its position is the scale reference point. If the scale has several – distance-coded – reference marks, then the scale reference point is the position of the leftmost reference mark (at the beginning of the measuring range).

Machine datum – miscellaneous function M91

The machine datum is required for the following tasks:

- Defining the limits of traverse (software limit switches)
- Moving to machine-referenced positions (such as tool change positions)
- Setting the workpiece datum

The distance for each axis from the scale reference point to the machine datum is defined by the machine manufacturer in a machine parameter.

If you want the coordinates in a positioning block to be referenced to the machine datum, end the block with M91.

Coordinates that are referenced to the machine datum are indicated in the display with REF.
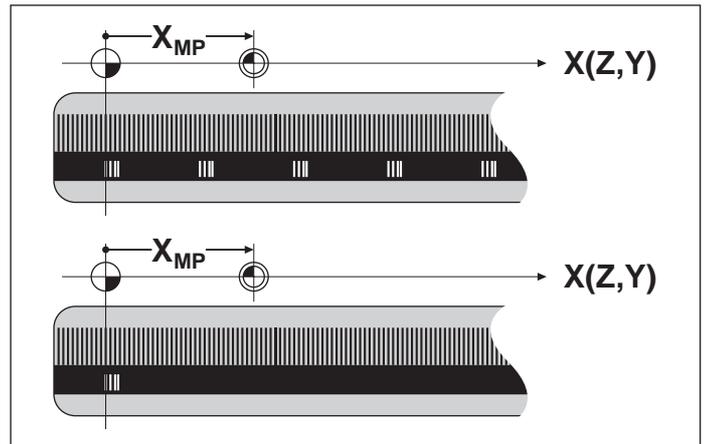
Fig. 5.48:   Scale reference point ⊕ and machine datum ⊕ on scales with one or more reference marks.

Additional machine datum – miscellaneous function M92

In addition to the machine datum, the machine manufacturer can also define an additional machine-based position as a reference point.

For each axis, the machine manufacturer defines the distance between the machine datum and this additional machine datum.

If you want the coordinates in a positioning block to be based on the additional machine datum, end the block with M92.

Radius compensation remains the same in blocks that are programmed with M91 or M92.

Workpiece datum

The user enters the coordinates of the datum for workpiece machining in the MANUAL OPERATION mode (see page 2-7).

If you want the coordinates to always be referenced to the machine datum or to the additional machine datum, you can inhibit datum setting for one or more axes.

If datum setting is inhibited for all axes, the TNC no longer displays the DATUM SET soft key in the MANUAL OPERATION mode.
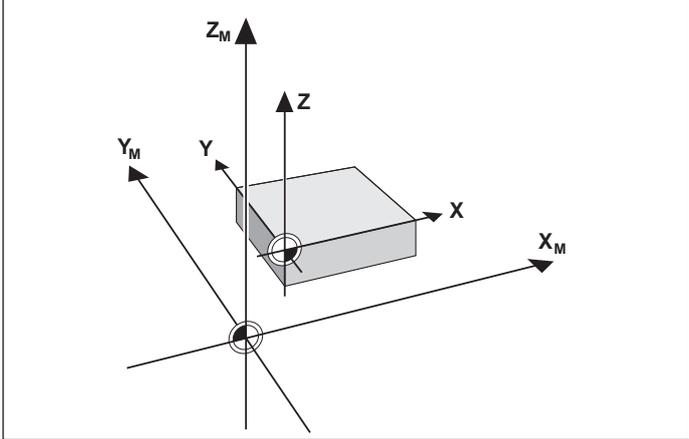
Fig. 5.49:   Machine datum ⊕ and workpiece datum ⊕

Feed rate factor for plunging movements: M103 F…

Standard behavior – without M103 F…

The TNC moves the tool at the last programmed feed rate, regardless of the direction of traverse.

Reducing the feed rate during plunging – with M103 F…

The TNC reduces the feed rate for movement in the negative direction of the tool axis to a given percentage of the last programmed feed rate:

$$F_{ZMAX} = F_{PROG} * F_\%$$

F_{ZMAX} :   Maximum feed rate in negative tool axis direction
F_{PROG} :   Last programmed feed rate
$F_\%$:   Programmed factor behind M103, in %

Cancelling

M103 F… is canceled by entering M103 without a factor.

Example

Feed rate for plunging is to be 20% of the feed rate in the plane

	Actual contouring feed rate [mm/min] with override 100%
.	
.	
.	
G01 G41 X+20 Y+20 F500 M103 F20	500
Y+50	500
G91 Z–2.5	100
Y+5 Z–5	367
X+50	500
G90 Z+5	500

☞   M103 F... is activated with machine parameter 7440 (see page 11-13).

Feed rate at circular arcs: M109/M110/M111

Standard behavior – M111

The programmed feed rate refers to the center of the tool path.

Constant contouring speed at circular arcs (feed rate increase and decrease) – M109

The TNC reduces the feed rate for circular arcs at inside contours such that the feed rate at the tool cutting edge remains constant. At outside contours the feed rate for circular arcs is correspondingly increased.

Constant contouring speed at circular arcs (feed rate decrease only) – M110

The TNC reduces the feed rate for circular arcs only at inside contours. At outside contours the feed rate remains the same.

Insert rounding arc between straight lines: M112 E...

Standard behavior – without M112 E...

A contour consisting of many short straight lines is normally machined such that the corners are cut as exactly as possible. When executing programs without tool radius compensation, the feed rate is therefore decreased to zero at corners.

Insert rounding arc between straight lines – with M112 E...

The TNC inserts a rounding arc between **two uncompensated straight lines**. The size of the arc depends on:

- the permissible deviation from the programmed contour entered for E (if no deviation is entered, the TNC assumes the permissible deviation to be infinite)
- the lengths of the two straight line segments between which the rounding arc is to be inserted
- the programmed feed rate (override setting 150%) and the circular acceleration (defined by the machine tool builder through machine parameters).

From these three criteria, the TNC calculates three rounding arcs and inserts the arc with the largest radius. If necessary, the contouring feed rate will automatically be reduced to comply with the programmed tolerance.

The permissible deviation E should be less than the programmed distance between points.

Fig. 5.50:   Permissible deviation E from the programmed contour

Parameter programming

You can also define E through Q parameters.

Duration of effect

M112 E... is effective during operation with feed precontrol as well as with servo lag. To cancel M112 E..., enter M113.

Automatic compensation of machine geometry when working with tilted axes: M114

Standard behavior – without M114

The TNC moves the tool to the positions given in the part program. The tool offset resulting from a tilted axis and the machine geometry must be calculated by a postprocessor.

Automatic compensation of machine geometry – with M114

The TNC compensates the tool offset resulting from positioning with tilted axes (such as dx and dz in Fig. 5.51). It calculates a 3D length compensation. The radius compensation must be calculated by a CAD system or by a postprocessor. A programmed radius compensation (RL or RR) results in the error message ILLEGAL NC BLOCK.

Thus if you write the NC program with a postprocessor, the machine geometry does not have to be calculated.

If the tool length compensation is calculated by the TNC, the programmed feed rate refers to the point of the tool; otherwise, it refers to the tool datum.
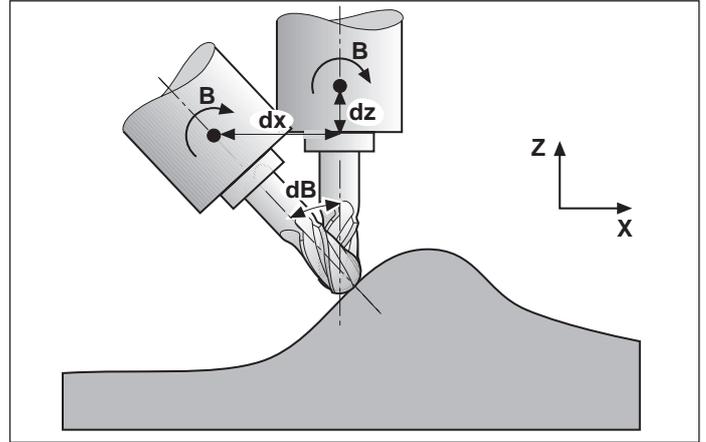
Fig. 5.51:   Offset of the tool datum for tilting the tool

If you are working with a swivel head under program control, you can interrupt program run and change the position of the tilt axis (for example, with the electronic handwheel.) Use the RESTORE POS AT N function (block scan, see page 3-8) to return to the point of interruption. The TNC automatically calculates the new tilt axis position.

Cancelling

M114 is cancelled by M115 or by a N99999 block.

The machine geometry must be defined by the machine manufacturer in machine parameters MP7510 and following.

Feed rate in mm/min on rotary axes A, B, C: M116

Standard behavior – without M116

The TNC interprets the programmed feed rate in a rotary axis in degrees per minute. The contouring feed rate therefore depends on the distance from the tool center to the center of the rotary axis. The larger this distance becomes, the greater the contouring feed rate.

Feed rate in mm/min on rotary axes – with M116

The TNC interprets the programmed feed rate in a rotary axis in mm/min. The contouring feed rate is therefore independent of the distance from the tool center to the center of the rotary axis.

Duration of effect

M116 is effective until the program ends (N99999 block), whereupon it is automatically cancelled.

The machine geometry must be entered in machine parameters 7510 ff. by the machine manufacturer.

Reduce display of a rotary axis to a value less than 360°: M94

Standard behavior – without M94

The TNC moves the tool from the current angular value to the programmed angular value.

Example: Current angular value: 538°
         Programmed angular value: 180°
         Actual path of traverse: −358°

Reduce display of rotary axis to value less than 360° – with M94

At the beginning of the block, the TNC first reduces the current angular value to a value less than 360° and then moves the tool to the programmed value. If several rotary axes are active, M94 will reduce the display of all rotary axes. To have the TNC reduce the display for a specific rotary axis only, enter the axis after M94.

Example: M94   Reduce display of all active rotary axes

         M94 C   Reduce display of the C axis only

         G00 C+180 M94   First reduce display of all active rotary axes, then move the tool in the C axis to the programmed value.
                                Current angular value: 538°
                                Programmed angular value: 180°
                                Actual path of traverse: +2°

Duration of effect

M94 is effective only at the beginning of the block in which it is programmed.

Optimized traverse of rotary axes: M126

Standard behavior – without M126

If you reduce display of a rotary axis to a value less than 360°, the TNC will move the axis in the following way:

Actual position	Nominal position	Actual path of traverse
350°	10°	–340°
10°	340°	+330°

Optimized traverse of rotary axes – with M126

If you reduce display of a rotary axis to a value less than 360°, the TNC will move the axis in the following way:

Actual position	Nominal position	Actual path of traverse
350°	10°	+20°
10°	340°	–30°

Resulting NC block: L C+10 A+340 R0 F500 M126

Duration of effect

M126 is effective at the beginning of the block. M126 is cancelled by M127 or at the end of program.

5.7 Positioning with Manual Data Input: System File $MDI

In the positioning with MDI mode you can program the system file $MDI.I
(or $MDI.H) for immediate execution. $MDI is programmed like any other
part program.

Applications

• Pre-positioning
• Face milling

To program the system file $MDI:

POSITIONING WITH MANL. DATA INPUT	
Select MDI operating mode.	

Program $MDI as desired.

To execute the system file $MDI:

POSITIONING WITH MANL. DATA INPUT	
Select POSITIONING MANUAL DATA INPUT operating mode.	

I	Start program run.

The system file $MDI must not contain a program call block (% block or cycle call).

Example application

Correcting workpiece misalignment on machines with rotary tables.

Make a basic rotation with the 3D touch probe, write down the
ROTATION ANGLE, then cancel the basic rotation again.

• Change the operating mode

| POSITIONING WITH MANL. DATA INPUT |
| Open the system file $MDI. |

• Program the rotation

G **0** **0**	• Select the rotary table axis. • Enter the ROTATION ANGLE you wrote down.
END	Confirm your entry.
I	The rotary axis corrects the misalignment.
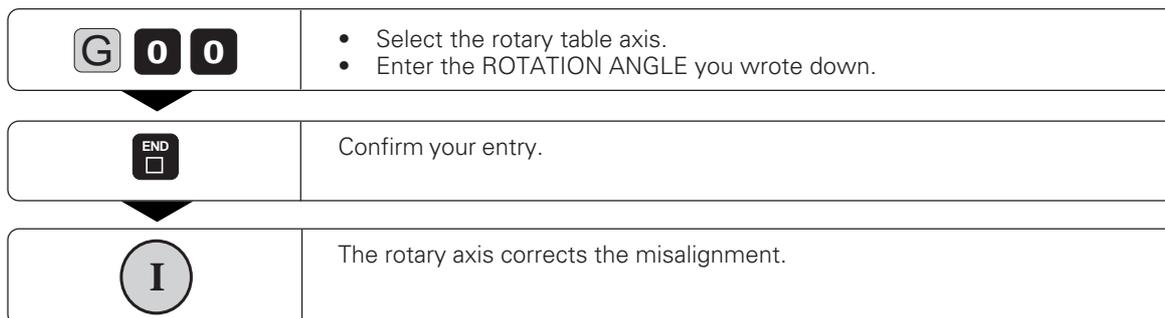
6   Subprograms and Program Section Repeats

Subprograms and program section repeats enable you to program a machining sequence once and then run it as often as desired.

Labels

Subprograms and program section repeats are marked by labels.

A label is identified by a number between 0 and 254. Each label (except label 0) can be set only once in a program. Labels are set with G98.

LABEL 0 marks the end of a subprogram.

6.1 Subprograms

Operating sequence

The main program is executed up to the block in which a subprogram is called with Ln,0 (①).

The subprogram is then executed from beginning to end (G98 L0) (②).

The main program is then resumed from the block after the subprogram call (③).
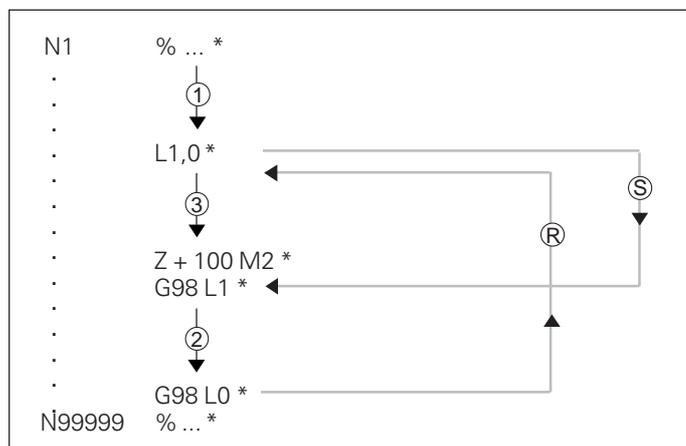
Fig. 6.1:   Flow diagram for subprogramming
Ⓢ= jump   Ⓡ= return jump

Operating limitations

- A main program can contain up to 254 subprograms.
- Subprograms can be called in any sequence and as often as desired.
- A subprogram cannot call itself.
- Subprograms should be written at the end of the main program (behind the block with M02 or M30).
- If subprograms are located before the block with M02 or M30, they will be executed at least once even if they are not called.

Programming and calling subprograms

Mark the beginning:

G 9 8 ENT	Select the label setting function.

LABEL NUMBER?	
5 END	The subprogram begins with (for example) label number 5.

Resulting NC block: G98 L5 *

Mark the end:

A subprogram always ends with label number 0.

G 9 8 ENT	Select the label setting function.

LABEL NUMBER?	
0 END	End of subprogram.

Resulting NC block: G98 L0 *

Call the subprogram:

A subprogram is called by its label number.

L 5 · 0 END	Call the subprogram behind label 5.

Resulting NC block: L5,0 *

	The command L0,0 is not permitted (label 0 is only used to mark the end of a subprogram).

Example for exercise: Group of four holes at three different locations

The holes are drilled with cycle G83 PECKING. Enter the setup clearance, feed rate, etc. in the cycle once. You can then call the cycle with miscellaneous function M99 (see page 8-3).

Coordinates of the first hole in each group:

Group ①   X = 15 mm   Y = 10 mm

Group ②   X = 45 mm   Y = 60 mm

Group ③   X = 75 mm   Y = 10 mm

Hole spacing: X = 20 mm
                        Y = 20 mm

Total hole depth: Z = 10 mm

Hole diameter: Ø = 5 mm
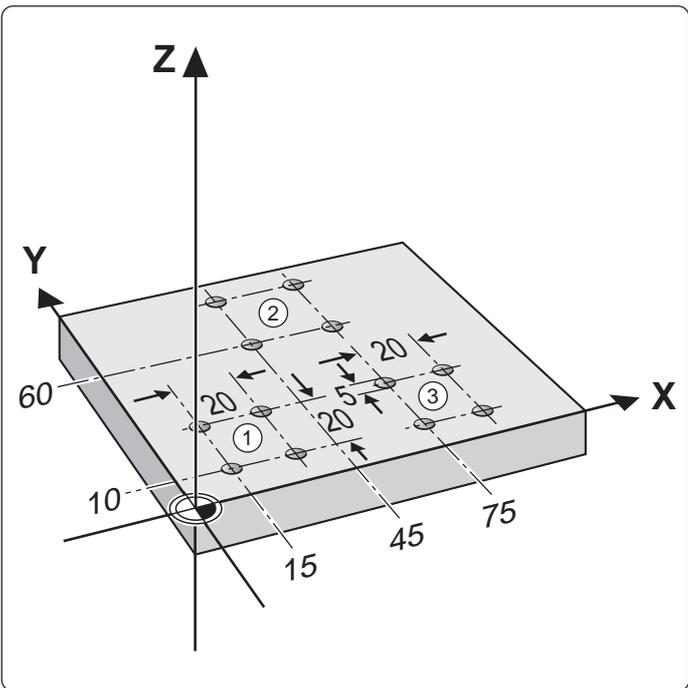
Part program

```
%S64I G71 * ............................................... Start program
N10 G30 G17 X+0 Y+0 Z–20 * ..................................... Define blank form
N20 G31 G90 X+100 Y+100 Z+0 *
N30 G99 T1 L+0 R+2.5 * ............................................. Define the tool
N40 T1 G17 S3500 * ..................................... Call the tool
N50 G83 P01 –2 P02 –10 P03 –5 P04 0
P05 100 * ........................................................ Cycle definition PECKING (see page 8-5)
N60 G00 G40 G90 Z+100 M06 * ................................... Retract and insert tool
N70 X+15 Y+10 * ....................................... Move to group 1
N80 Z+2 M03 * ........................................... Pre-position in the infeed axis
N90 L1,0 * ................................................. Call subprogram (subprogram executed with block N90)
N100 X+45 Y+60 * ..................................... Move to group 2
N110 L1,0 * ................................................. Call subprogram
N120 X+75 Y+10 * ..................................... Move to group 3
N130 L1,0 * ................................................. Call subprogram
N140 Z+100 M02 * ..................................... Retract in the infeed axis;
                                                       end of main program (M02); the subprogram is entered
                                                       behind M02
N150 G98 L1 * ............................................ Beginning of subprogram
N160 G79 * ................................................. Perform pecking cycle for first hole
N170 G91 X+20 M99 * ................................. Move to second hole (incremental) and drill
N180 Y+20 M99 * ....................................... Move to third hole (incremental) and drill
N190 X–20 G90 M99 * ................................. Move to fourth hole (incremental) and drill; change to
                                                       absolute coordinates (G90)
N200 G98 L0 * ............................................ End of subprogram
N99999 %S64I G71 * .................................... End of program
```

6.2 Program Section Repeats

Like subprograms, program section repeats are identified with labels.

Operating sequence

The program is executed up to the end of the labelled program section (① and ②), i.e. up to the block with Ln,m.

Then the program section between the called label and the label call is repeated the number of times entered after under m (③, ④).

The program is then resumed after the last repetition (⑤).

Programming notes

- A program section can be repeated up to 65 534 times in succession.
- The total number of times the program section is executed is always one more than the programmed number of repeats.
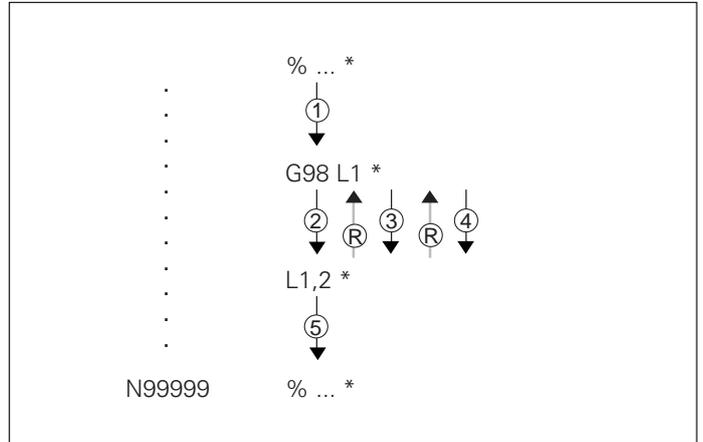
Fig. 6.2:   Flow diagram for a program section repeat;
ⓡ = return jump

Programming and executing a program section repeat

Mark the beginning:

[G] [9] [8] [ENT]	Select the label setting function.

LABEL NUMBER ?	
[7] [END]	Program section repeated starting at LABEL 7, for example.

Resulting NC block: G98 L7 *

Specify the number of repeats:

Enter the number of repeats in the block that calls the label. This is also the block that ends the program section.

[L] [7] [·] [1] [0] [END]	The program section from LABEL 7 up to this block will be repeated ten times. This means it will be run a total of *eleven times*.

Resulting NC block: L7,10 *

Example for exercise: Row of holes parallel to the X axis

Coordinates of the first hole:	X	=	5 mm
	Y	=	10 mm
Hole spacing:	IX	=	15 mm
Number of holes:	N	=	6
Depth:	Z	=	10
Hole diameter:	Ø	=	5 mm
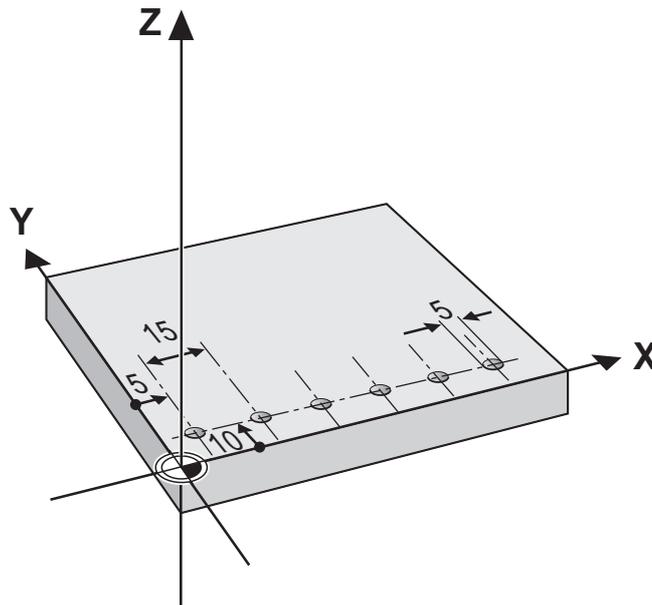
Part program

%S66I G71 *	Start program
N10 G30 G17 X+0 Y+0 Z–20 *	Define blank form
N20 G31 G90 X+100 Y+100 Z+0 *	
N30 G99 T1 L+0 R+2.5 *	Define tool
N40 T1 G17 S3500 *	Call tool
N50 G00 G40 G90 Z+100 M06 *	Retract and insert tool
N60 X–10 Y+10 Z+2 M03 *	Pre-position to the point which is offset in negative X direction by the hole spacing
N70 G98 L1 *	Start of the program section to be repeated
N80 G91 X+15 *	Move to drilling position (incremental dimension)
N90 G01 G90 Z–10 F100 *	Drill (absolute dimension)
N100 G00 Z+2 *	Retract
N110 L1,5 *	Call LABEL 1; repeat program section from block N70 to block N110 five times (total of six holes)
N120 Z+100 M02 *	Retract in the infeed axis
N99999 %S66I G71 *	

Example for exercise: Milling without radius compensation using program section repeats

Sequence:

- Upward milling direction

- Machine the area from X=0 to 50 mm (program all X coordinates with the tool radius *subtracted*) and from Y=0 to 100 mm: G98 L1

- Machine the area from X=50 to X=100 mm (program all X coordinates with the tool radius *added*) and from Y=0 to 100 mm: G98 L2

- After each upward pass, the tool is moved by an increment of +2.5 mm in the Y axis.

The illustration at right shows the block numbers containing the end points of the corresponding contour elements.
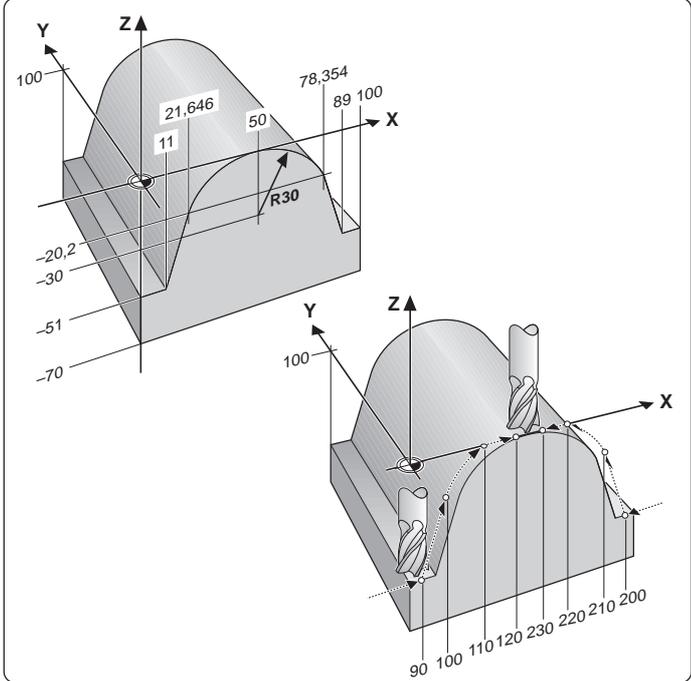
Part program

```
%S67I G71 * ............................................. Start program
N10 G30 G17 X+0 Y+0 Z–70 * ................................ Define blank form (note new values)
N20 G31 G90 X+100 Y+100 Z+0 *
N30 G99 T1 L+0 R+10 * ......................................... Define tool
N40 T1 G17 S1750 * .................................................. Call tool
N50 G00 G40 G90 Z+100 M06 * ............................. Retract and insert tool
N60 X–20 Y–1 M03 * ................................................ Pre-position in the plane
N70 G98 L1 * ......................................................... Start of program section 1
N80 G90 Z–51 *
N90 G01 X+1 F100 *
N100 X+11.646 Z–20.2 * ......................................... Program section for machining from
N110 G06 X+40 Z+0 * ............................................. X = 0 to 50 mm and Y = 0 to 100 mm
N120 G01 X+41 *
N130 G00 Z+10 *
N140 X–20 G91 Y+2.5 *
N150 L1,40 * .......................................................... Call LABEL 1, repeat program section from block
                                                                        N70 to N150 forty times
N160 G90 Z+20 * .................................................... Retract in the infeed axis
N170 X+120 Y–1 * ................................................... Pre-position for program section 2
N180 G98 L2 * ....................................................... Start of program section 2
N190 G90 Z–51 *
N200 G01 X+99 F100 *
N210 X+88.354 Z–20.2 * ......................................... Program section for machining from
N220 G06 X+60 Z+0 * ............................................. X = 50 to 100 mm and Y = 0 to 100 mm
N230 G01 X+59 *
N240 G00 Z+10 *
N250 X+120 G91 Y+2.5 *
N260 L2,40 * .......................................................... Call LABEL 2, repeat program section from block
                                                                        N180 to N260 forty times
N270 G90 Z+100 M02 * ........................................... Retract in the infeed axis
N99999 %S67I G71 *
```

6.3 Program as Subprogram

Operating sequence

A program is executed (①) up to the block in which another program is called (block with %).

Then the other program is run from beginning to end (②).

The first program is then resumed beginning with the block behind the program call (③).

Operating limitations

- Programs called from an external data medium (e.g., floppy disk) must not contain any subprograms or program section repeats.
- No labels are needed to call main programs as subprograms.
- The called program must not contain the miscellaneous functions M02 or M30.
- The called program must not contain a jump into the calling program.
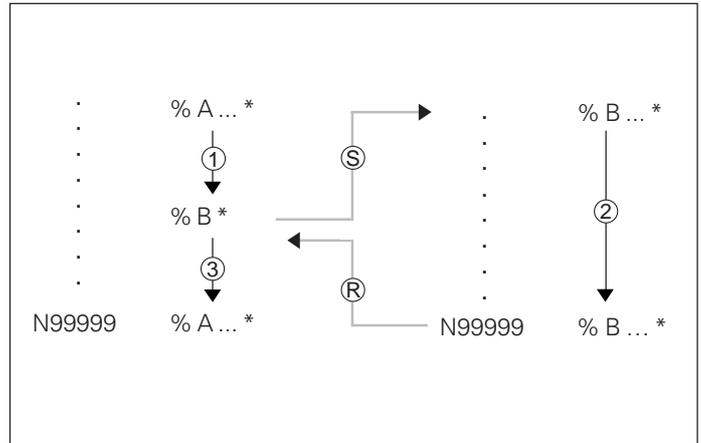
Fig. 6.3:   Flow diagram of a program as subprogram.
ⓈＳ = jump, Ⓡ = return jump

Calling a program as a subprogram

PROGRAM NAME ?

Enter the name of the program that you wish to call from this block.

Function	Soft key
Call a plain-language program	.H
Call an ISO program	.I
Call an externally stored program	EXT

Resulting NC block: % NAME

- You can also call a program with cycle G39 (see page 8-53).
- When calling an ISO program, the program name must not contain G50, G70 or G71.

The following additional notes apply to the TNC 426:
- If the program you want to call is located in the same directory as the program you are calling it from, you only have to enter the program name.
- If the program you want to call is not located in the same directory as the program you are calling it from, then you must enter the full path (e.g., \VZW35\ROUGH\PGM1.I).
- If you want to call a plain-language program, enter the file type .H behind the program name.

6.4 Nesting

Subprograms and program section repeats can be nested in the following ways:

- Subprograms within a subprogram
- Program section repeats within a program section repeat
- Subprograms repeated
- Program section repeats within a subprogram

Nesting depth

The nesting depth is the number of successive levels in which program sections or subprograms can call further program sections or subprograms.

Maximum nesting depth for subprograms: 8

Maximum nesting depth for calling main programs: 4

Subprogram within a subprogram

Program layout

```
                    % UPGMS G71 *
          .           │
          .          ① │
          .           ▼
e.g. N17   L1,0 * ...................................... Call subprogram at G98 L1
          .           │
          .          ⑤ │
          .           ▼
e.g. N35   G00 G40 Z+100 M2 * ............................ Last block of main program (with M2)
```

```
      N36   G98 L1 *                                              ⎫
       .           │                                              │
       .          ② │                                            │   Subprogram 1
       .           ▼                                             ⎬   with call of
e.g. N39   L2,0 *                                                │   subprogram 2
       .           │                                              │
       .          ④ │                                            │
       .           ▼                                             │
e.g. N45   G98 L0 * .................................... End of subprogram 1   ⎭
```

```
      N46   G98 L2 *                                              ⎫
       .           │                                              │
       .          ③ │                                            ⎬   Subprogram 2
       .           ▼                                             │
e.g. N62   G98 L0 * .................................... End of subprogram 2   ⎭
```

```
  N99999   % UPGMS G71 * .................................... End of main program
```
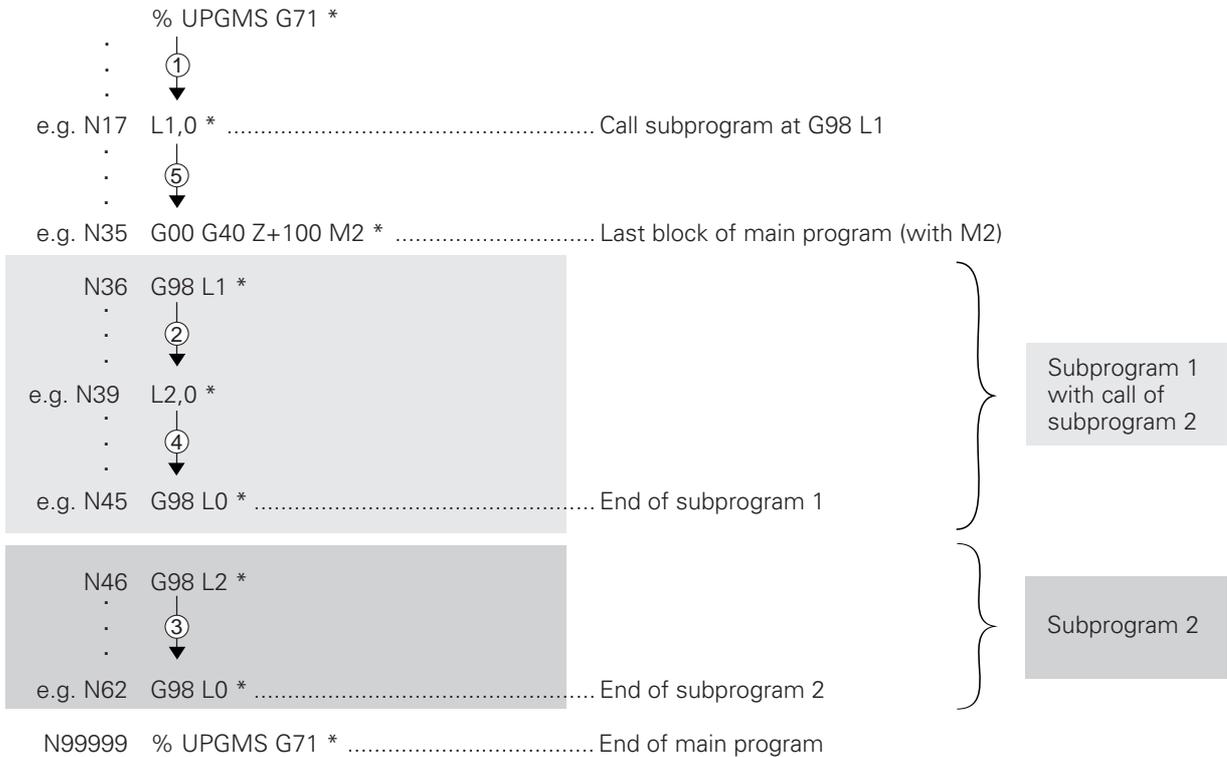
Program execution

1st step:	The main program UPGMS is executed up to block 17.
2nd step:	Subprogram 1 is called, and executed up to block 39.
3rd step:	Subprogram 2 is called, and executed up to block 62. End of subprogram 2 and return jump to the subprogram from which it was called.
4th step:	Subprogram 1 is called, and executed from block 40 to block 45. End of subprogram 1 and return jump to the main program UPGMS.
5th step:	Main program UPGMS is executed from block 18 to block 35. Return jump to block 1 and end of program.

Example for exercise: Three groups of four holes (see page 6-4) with three different tools

Machining sequence:

Countersinking – Drilling – Tapping

Machining data is entered in cycle G83: PECK DRILLING (see page 8-4) and cycle G84: TAPPING (see page 8-6). The tool moves to the hole groups in a subprogram, while the machining is performed in a second subprogram.

Coordinates of the first hole in each group:

① X = 15 mm   Y = 10 mm

② X = 45 mm   Y = 60 mm

③ X = 75 mm   Y = 10 mm

Hole spacing: IX = 20 mm   IY = 20 mm

Hole data:
Countersinking   ZC =   3 mm   Ø = 7 mm
Drilling   ZD = 15 mm   Ø = 5 mm
Tapping   ZT = 10 mm   Ø = 6 mm
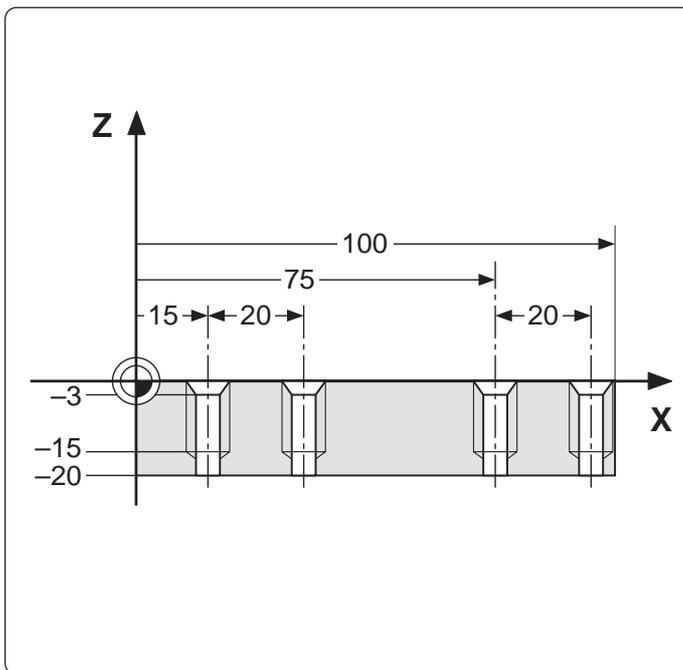
Part program

```
%S610I G71 * ........................................... Start program
N10 G30 G17 X+0 Y+0 Z–20 * ................................ Define bank form
N20 G31 G90 X+100 Y+100 Z+0 *
N30 G99 T25 L+0 R+2.5 * ....................................... Tool definition for pecking
N40 G99 T30 L+0 R+3 * .......................................... Tool definition for countersinking
N50 G99 T35 L+0 R+3.5 * ....................................... Tool definition for tapping
N60 T35 G17 S3000 * ............................................. Tool call for countersinking
N70 G83 P01 –2 P02 –3 P03 –3 P04 0
P05 100 * ............................................................... Cycle definition pecking
N80 L1,0 * ............................................................. Call subprogram 1
N90 T25 G17 S2500 * ............................................. Tool call for pecking
N100 G83 P01 –2 P02 –25 P03 –10 P04 0
P05 150 * ............................................................... Cycle definition pecking
N110 L1,0 * ........................................................... Call subprogram 1
N120 T30 G17 S100 * ............................................. Tool call for tapping
N130 G84 P01 –2 P02 –15 P03 0.1 P04 100 * ........ Cycle definition tapping
N140 L1,0 * ........................................................... Call subprogram 1
N150 Z+100 M02 * .................................................. Retract in the infeed axis; end of main program
N160 G98 L1 * ........................................................ Start subprogram 1
N170 G00 G40 G90 X+15 Y+10 M03 * .................... Move to hole group 1
N180 Z+2 * .............................................................. Pre-position in the infeed axis
N190 L2,0 * ............................................................ Call subprogram 2
N200 X+45 Y+60 * .................................................. Move to hole group 2
N210 L2,0 * ............................................................ Call subprogram 2
N220 X+75 Y+10 * .................................................. Move to hole group 3
N230 L2,0 * ............................................................ Call subprogram 2
N240 G98 L0 * ........................................................ End of subprogram 1

N250 G98 L2 * ........................................................ Start of subprogram 2
N260 G79 *
N270 G91 X+20 M99 * ............................................ Drill holes with currently active cycle
N280 Y+20 M99 *
N290 X–20 G90 M99 *
N300 G98 L0 * ........................................................ End of subprogram 2
N99999 %S610I G71 *
```

Repeating program section repeats

Program layout

```
              % REPS G71 *
    .
    .           ①
    .           ↓

e.g. N15    G98 L1 *
    .           ①   1x ④
    .               ↓

e.g. N20    G98 L2 *
    .           ①  2x ②  1x ④  2x ⑤
    .               ↓      ↓      ↓

e.g. N27    L2,2 * ..................................... The program section between this block and G98 L2 (block 20)
    .                                                    is repeated twice
    .           ③      ⑥
    .               ↓

e.g. N35    L1,1 * ..................................... The program section between this block and G98 L1 (block 15)
    .                                                    is repeated once
    .           ⑦
    .           ↓
    .

N99999      % REPS G71 *
```
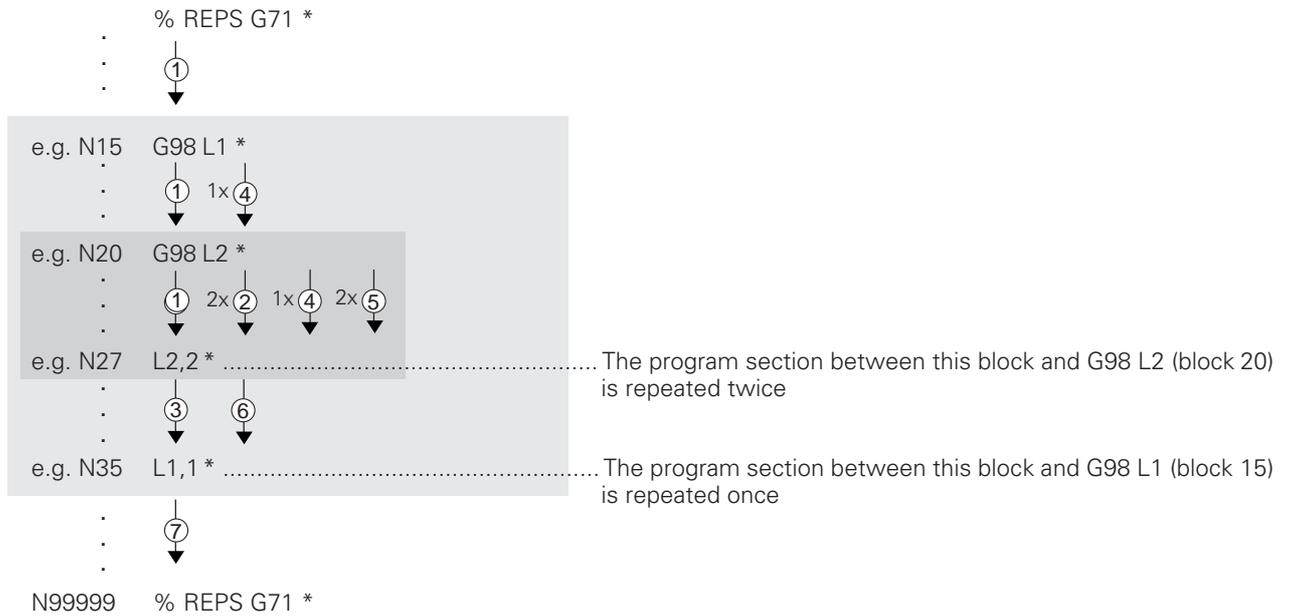
Program execution

1st step: Main program REPS is executed up to block 27.

2nd step: Program section between block 27 and block 20 is repeated twice.

3rd step: Main program REPS is executed from block 28 to block 35.

4th step: Program section between block 35 and block 15 is repeated once.

5th step: Repetition of the second step within step ④.

6th step: Repetition of the third step within step ④.

7th step: Main program REPS is executed from block 36 to block 50. End of program.

Repeating subprograms

Program structure

```
                     % UPGREP G71 *
        .
        .                    ┃
        .                    ①
                             ▼
e.g.   N10    G98 L1 *
       N11    L2,0 * ....................2x ③................. Call subprogram
       N12    L1,2 * ........................▼................. Program section repeat
        .             ④
        .             ▼
        .
e.g.   N19    G00 G40 Z+100 M2 * ........................ Last program block of main program with M2
       N20    G98 L2 * ............................ Start of subprogram
        .
        .              ②
        .              ▼
e.g.   N28    G98 L0 * ............................ End of subprogram
       N99999 % UPGREP G71 * .............................. End of main program
```

Program execution

1st step: Main program UPGREP is executed up to block 11.

2nd step: Subprogram 2 is called and executed.

3rd step: Program section from block 12 to block 10 is repeated twice,
           so subprogram 2 is repeated twice.

4th step: Main program UPGREP is executed from block 13 to block 19. End of program.

7   Programming with Q Parameters

Q Parameters are used for:

- Programming families of parts

- Defining contours through mathematical functions

An entire **family of parts** can be programmed on the TNC with **a single part program**. You do this by entering variables called *Q parameters* instead of fixed numerical values.

Q parameters are designated by the letter Q and a number between 0 and 299.

Fig. 7.1: Q parameters as variables

Meaning	Range
Freely applicable parameters, locally effective (depending on MP7251)	Q0 to Q99
Parameters for special functions of the TNC	Q100 to Q199
Additional parameters, only available on TNC 426: Parameters that are primarily used for cycles, globally effective	Q200 to Q299

Q parameters can represent information such as:
- coordinate values
- feed rates
- rpm
- cycle data

Q parameters also enable you to program **contours** that are defined through **mathematical functions**.

In addition, you can use Q parameters to make the execution of machining steps depend on certain **logical conditions**.

You can **mix Q parameters** and **fixed numerical values** within a program.

Q parameters can be assigned numerical values between –99999.9999 and +99999.9999.

You can enter the individual Q parameter functions either blockwise (see pages 7-3 to 7-12) or together in a formula through the ASCII keyboard (see pages 7-20 to 7-22).

☞   Certain Q parameters are always assigned the same data by the TNC. For example, Q108 is always assigned the current tool radius. A list of these parameters can be found in Chapter 11.

Use the Q key (below the -/+ key) to select the Q parameter functions. The following soft keys appear, with which you can select function groups:

BASIC ARITH- METIC	TRIGO- NOMETRY	JUMP	DIVERSE FUNCTION	FORMULA			END

Function	Soft key
Basic arithmetic (assign, add, subtract, multiply, divide, square root)	BASIC ARITH- METIC
Trigonometric functions	TRIGO- NOMETRY
If/Then conditions, jumps	JUMP
Other functions	DIVERSE FUNCTION
Enter formula directly from keyboard	FORMULA

7.1 Part Families — Q Parameters in Place of Numerical Values

The Q parameter function D0: ASSIGN assigns numerical values to
Q parameters.
Example: Q10 = 25

This enables you to use variables in the program instead of fixed
numerical values.
Example: X + Q10 (= X + 25)

You only need to write one program for a whole
family of parts, entering the characteristic dimen-
sions as Q parameters. To program a particular part,
you then assign the appropriate values to the
individual Q parameters.

Example

Cylinder with Q parameters

Radius	R	=	Q1
Height	H	=	Q2
Cylinder Z1:	Q1	=	+30
	Q2	=	+10
Cylinder Z2:	Q1	=	+10
	Q2	=	+50
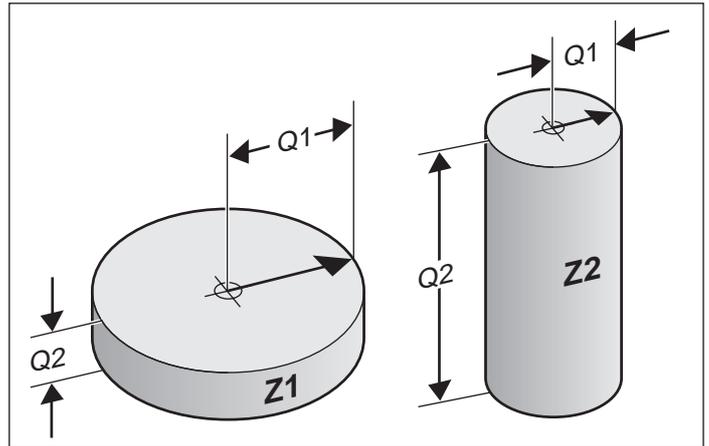
Fig. 7.2: Part dimensions as Q parameters

To assign numerical values to Q parameters:

PARA- METER	Select PARAMETER.
BASIC ARITH- METIC	Select BASIC ARITHMETIC.
D0 X = Y	Select D0: ASSIGN.
PARAMETER NUMBER FOR RESULT ?	
5 ENT	Enter the Q parameter number, for example 5.
FIRST VALUE / PARAMETER ?	
6 END	Enter a value or another Q parameter whose value is to be assigned to Q5.

Resulting NC block: D00 Q5 P01 +6*

Example for exercise: Full circle

Circle center I, J:	X	=	50 mm
	Y	=	50 mm
Beginning and end of circular arc C:	X	=	50 mm
	Y	=	0 mm
Milling depth:	Z_M	=	–5 mm
Tool radius:	R	=	15 mm
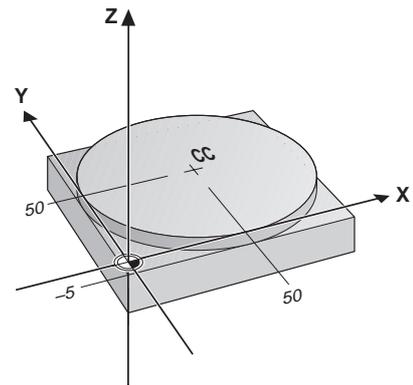
Part program without Q parameters

%S520I G71 *	Start of program
N10 G30 G17 X+1 Y+1 Z–20 *	Blank form MIN point
N20 G31 G90 X+100 Y+100 Z+0 *	Blank form MAX point
N30 G99 T6 L+0 R+15 *	Define tool
N40 T6 G17 S1500 *	Call tool
N50 G00 G40 G90 Z+100 M06 *	Retract and insert tool
N60 X+50 Y–40 *	Pre-position in the working plane
N70 Z5 M03 *	Move tool to working depth
N80 I+50 J+50 *	Coordinates of the circle center
N90 G01 G41 X+50 Y+0 F100 *	Move to first contour point with radius compensation at machining feed rate
N100 G26 R10 *	Soft (tangential) approach
N110 G02 X+50 Y+0 *	Mill arc around circle center I, J; negative rotation; coordinates of end point X = +50 mm and Y = +0
N120 G27 R10 *	Soft (tangential) departure
N130 G00 G40 X+50 Y–40 *	Depart contour, cancel radius compensation
N140 Z+100 M02 *	Retract in the infeed axis
N99999 %S520I G71 *	

Part program with Q parameters

%S74I G71 *	Start of program
N10 D00 Q1 P01 +100 *	Clearance height
N20 D00 Q2 P01 +30 *	Start position X
N30 D00 Q3 P01 –20 *	Start/end position Y
N40 D00 Q4 P01 +70 *	End position X
N50 D00 Q5 P01 –5 *	Milling depth
N60 D00 Q6 P01 +50 *	Circle center X
N70 D00 Q7 P01 +50 *	Circle center Y
N80 D00 Q8 P01 +50 *	Circle starting point X
N90 D00 Q9 P01 +0 *	Circle starting point Y
N100 D00 Q10 P01 +0 *	Tool length L
N110 D00 Q11 P01 +15 *	Tool radius R
N120 D00 Q20 P01 +100 *	Milling feed rate F
N130 G30 G17 X+1 Y+1 Z–20 *	
N140 G31 G90 X+100 Y+100 Z+0 *	
N150 G99 T6 L+Q10 R+Q11 *	
N160 T6 G17 S1000 *	
N170 G00 G40 G90 Z+Q1 M06 *	
N180 X+Q2 Y+Q3 *	
N190 Z+Q5 M03 *	Block N130 to N260 correspondingly
N200 I+Q6 J+Q7 *	Block N10 to N140 from program S520I.I
N210 G01 G41 X+Q8 Y+Q9 FQ20 *	
N220 G26 R10 *	
N230 G02 X+Q8 Y+Q9 *	
N240 G27 R10 *	
N250 G00 G40 X+Q4 Y+Q3 *	
N260 Z+Q1 M02 *	
N99999 %S74I G71 *	

7.2 Describing Contours Through Mathematical Functions

Select the BASIC ARITHMETIC soft key to call the following functions:

D0 X = Y	D1 X + Y	D2 X - Y	D3 X * Y	D4 X / Y	D5 SQRT		END

Overview

The mathematical functions assign the result of one of the following operations to a Q parameter:

	Soft key
D0: ASSIGN Example: D00 Q5 P01 +60∗ Assigns a numerical value.	D0 X = Y
D1: ADDITION Example: D01 Q1 P01 −Q2 P02 −5∗ Calculates and assigns the sum of two values.	D1 X + Y
D2: SUBTRACTION Example: D02 Q1 P01 +10 P02 +5∗ Calculates and assigns the difference of two values.	D2 X - Y
D3: MULTIPLICATION Example: D03 Q2 P01 +3 P02 +3∗ Calculates and assigns the product of two values.	D3 X * Y
D4: DIVISION Example: D04 Q4 P01 +8 P02 +Q2∗ Calculates and assigns the quotient of two values. **Not permitted: division by 0**	D4 X / Y
D5: SQUARE ROOT Example: D05 Q20 P01 +4∗ Calculates and assigns the square root of a number. **Not permitted: square root of a negative number**	D5 SQRT

In the above table, "values" can be any of the following:

- two numbers
- two Q parameters
- a number and a Q parameter

The Q parameters and numerical values in the equations can be entered with positive or negative signs.

7.2   Describing Contours Through Mathematical Functions

Programming example for basic arithmetical operations

Assign the value 10 to parameter Q5, and assign the product of Q5 and
the value 7 to Q12.

PARA- METER	Select PARAMETER.

BASIC ARITH- METIC	Select BASIC ARITHMETIC.

D0 X = Y	Select function D0: ASSIGN.

PARAMETER NUMBER FOR RESULT ?

5 ENT	Enter parameter number, for example Q5.

FIRST VALUE / PARAMETER ?

1 **0** END	Assign numerical value to parameter Q5.

7.2   Describing Contours Through Mathematical Functions

PARA-METER	Select PARAMETER.
BASIC ARITH-METIC	Select BASIC ARITHMETIC.
D3 X * Y	Select function D3: MULTIPLICATION.

PARAMETER NUMBER FOR RESULT ?

| **1** **2** ENT | Enter parameter number, for example Q12. |

FIRST VALUE/PARAMETER ?

| **Q** **5** ENT | Enter Q5 (=10) and confirm. |

SECOND VALUE/PARAMETER?

| **7** END | Enter 7 and confirm. |

Resulting NC blocks: *D00 Q5   P01 +10*
                         D03 Q12 P01 +Q5 D02 +7

7.3 Trigonometric Functions

Sine, *cosine* and *tangent* are terms designating the ratios of the sides of right triangles.

For a right triangle, the trigonometric functions of the angle α are defined by the equations

$$\sin \alpha = a/c,$$
$$\cos \alpha = b/c,$$
$$\tan \alpha = a/b = \sin \alpha / \cos \alpha,$$

where

- *c* is the side opposite the right angle
- *a* is the side opposite angle α
- *b* the third side.

The angle can be found from the tangent:

$$\alpha = \text{arc tan } \alpha = \text{arc tan } (a/b) = \text{arc tan } (\sin \alpha / \cos \alpha)$$

Example: *a* = 10 mm
          b = 10 mm
          α = arc tan (*a* / *b*) = arc tan 1 = 45°

Furthermore, $a^2 + b^2 = c^2$ ($a^2 = a \cdot a$)
               $c = \sqrt{a^2 + b^2}$

Fig. 7.3: Sides and angles on a right triangle

Select the trigonometric functions to call the following options:

D6 SIN(X)	D7 COS(X)	D8 X LEN Y	D13 X ANG Y				END

Overview

	Soft key
D6: SINE Example: D06 Q20 P01 –Q5 ∗ Calculate the sine of an angle in degrees (°) and assign it to a parameter	D6 SIN(X)
D7: COSINE Example: D07 Q21 P01 –Q5 ∗ Calculate the cosine of an angle in degrees (°) and assign it to a parameter	D7 COS(X)
D8: ROOT-SUM OF SQUARES Example: D08 Q10 P01 +5 P02 +4 ∗ Take the square root of the sum of two squared numbers and assign it to a parameter	D8 X LEN Y
D13: ANGLE Example: D13 Q20 P01 +10 P02 –Q1∗ Calculate the angle from the arc tangent of two sides or from the sine and cosine of the angle ($0 \leq$ angle $\leq 360°$) and assign it to a parameter	D13 X ANG Y

7.4 If-Then Decisions with Q Parameters

The TNC can make logical If-Then decisions by comparing a Q parameter with another Q parameter or with a numerical value.

Jumps

The jump target is specified by a label number in the decision block. If the programmed condition is fulfilled, the TNC continues the program at the specified label. If it is not fulfilled, it continues with the next block.

To jump to another program, enter a program call with % (see page 6-8) after the block with the target label.

Unconditional jumps

An unconditional jump is programmed by entering a conditional jump whose condition is always true. Example:

> If 10 equals 10, go to label 1
>
> D09 P01 +10 P02 +10 P03 1

Select the jump function to display the following options:

D9 IF X EQ Y GOTO	D10 IF X NE Y GOTO	D11 IF X GT Y GOTO	D12 IF X LT Y GOTO				END

Overview

	Soft key
D9: IF EQUAL, JUMP Example: D09 P01 +Q1 P02 +Q3 P03 5 ∗ If the two values or parameters are equal, jump to the given label.	D9 IF X EQ Y GOTO
D10: IF NOT EQUAL, JUMP Example: D10 P01 +10 P02 −Q5 P03 10 ∗ If the two values or parameters are not equal, jump to the given label.	D10 IF X NE Y GOTO
D11: IF GREATER THAN, JUMP Example: D11 P01 +Q1 P02 +10 P03 5 ∗ If the first value or parameter is greater than the second value or parameter, jump to the given label.	D11 IF X GT Y GOTO
D12: IF LESS THAN, JUMP Example: D12 P01 +Q5 P02 +0 P03 1 ∗ If the first value or parameter is less than the second value or parameter, jump to the given label.	D12 IF X LT Y GOTO

Jump example

You want to jump to program 100.H as soon as Q5 becomes negative.

```
        .
        .
        .
N5      D00 Q5 P01 +10 * ........................................... Assign a value, such as +10, to parameter Q5
        .
        .
        .
N9      D02 Q5 P01 +Q5 P02 +12 * ........................... Reduce the value of Q5
N10     D12 P01 +Q5 P02 +0 P03 5 * ......................... If +Q5 < 0, jump to label 5
        .
        .
        .
N15     G98 L5 * ....................................................... Label 5
N16     % 100.H * ..................................................... Jump to program 100.H
        .
        .
        .
```

7.5 Checking and Changing Q Parameters

During a program run or program test, Q parameters can be checked and changed if necessary.

Preparation:

• If you are in a program run, interrupt it (for example by pressing the machine STOP key and the INTERNAL STOP soft key)
• If you are doing a test run, interrupt it

To call the Q parameter:

Q ▶	**Q** ___ **=**	
	1 0 ENT	Select the parameter, for example Q10.

Q10 = + 100
The TNC displays the current value.

0 ENT	Change the Q parameter, for example Q10 = 0.
ENT	Leave the Q parameter unchanged.

7.6 Diverse Functions

Select the diverse functions to call the following options for the TNC 425, TNC 415 B and TNC 407:

D14 ERROR=	D15 PRINT	D19 PLC=					END

Select the diverse functions to call the following options for the TNC 426:

D14 ERROR=	D15 PRINT	D16 F-PRINT	D17 SYS-DATUM WRITE	D18 SYS-DATUM READ	D19 PLC=		

Displaying error messages

D14 ERROR=

With the function D14: ERROR you can call messages that were preprogrammed by the machine tool builder.

If the TNC encounters a block with D14 during a program run or test run, it will interrupt the run and display an error message. The program must then be restarted.

Input

Example: D14 P01 254

The TNC then displays the text stored under error number 254.

Error number to be entered	Prepared dialog text
0 to 299	D14: ERROR CODE 0 299
300 to 399	PLC: ERROR 0 ... 99
400 to 499	CYCLE PARAMETER 0 99
300 to 999 *(TNC 426 only)*	No standard dialog text prepared
1000 to 1099 *(TNC 426 only)*	Internal error message (see table on next page)

Your machine tool builder may have programmed a dialog text that differs from the above.

7.6   Diverse Functions

Error code	Error text
1000	SPINDLE MUST BE TURNING
1001	TOOL AXIS IS MISSING
1002	SLOT WIDTH TOOL LARGE
1003	TOOL RADIUS TOO LARGE
1004	RANGE EXCEEDED
1005	START POSITION INCORRECT
1006	ROTATION NOT PERMITTED
1007	SCALING FACTOR NOT PERMITTED
1008	MIRRORING NOT PERMITTED
1009	DATUM SHIFT NOT PERMITTED
1010	FEED RATE IS MISSING
1011	ENTRY VALUE INCORRECT
1012	WRONG SIGN PROGRAMMED
1013	ENTERED ANGLE NOT PERMITTED
1014	TOUCH POINT INACCESSIBLE
1015	TOO MANY POINTS
1016	CONTRADICTORY ENTRY
1017	CYCL INCOMPLETE
1018	PLANE WRONGLY DEFINED
1019	WRONG AXIS PROGRAMMED
1020	WRONG RPM
1021	RADIUS COMP. UNDEFINED
1022	ROUNDING-OFF UNDEFINED
1023	ROUNDING RADIUS TOO LARGE
1024	PROGRAM START UNDEFINED
1025	EXCESSIVE SUBPROGRAMMING
1026	ANGLE REFERENCE MISSING

Output through an external data interface

```
  D15
PRINT
```

The function D15: PRINT transfers the values of Q parameters and error messages through the data interface, for example to a printer.

- D15: PRINT with numerical values up to 200
  Example: D15: PRINT 20
  Transfers the corresponding error message (see overview for D14).

- D15: PRINT with Q parameter
  Example: D15: PRINT Q20
  Transfers the value of the corresponding Q parameter.

You can transfer up to six Q parameters and numerical values simultaneously.
Example: D15 P01 1 P02 Q1 P03 2 P04 Q2

The following notes apply to TNC 407, TNC 415 B and TNC 425 controls:
- If the part program is interrupted while D15 is active, you must close the interface with the CLOSE RS-232-C soft key *(only available on TNC 407, TNC 415 B and TNC 425)*.
- When D15 is used for transferring values to a PC, the TNC generates the file %FN15RUN.A in the PC memory to store the transferred values *(only available on TNC 407, TNC 415 B and TNC 425)*.

The following note applies to TNC 426 controls:
- The path for storing the texts and Q parameter values is entered in the menu option PRINT or PRINT TEST (see "Setting the Data Interface").

Formatted output of texts and Q parameter values

```
    D16
  F -PRINT
```

407

415

425

👆 The path for storing the texts and Q parameter values is entered in the menu option PRINT or PRINT TEST (see "Setting the Data Interface", page 10-5).

The function D16: F-PRINT transfers texts and Q parameter values in a selectable format through the data interface, for example to a printer. When you store the values in the TNC memory or transfer them to a PC, the TNC stores the data in the file %D16RUN.A (output in program run mode) or in the file %D16SIM.A (output in test run mode). You can define the output format by programming a text file.

Example D16 P01 TNC:\MASK\MASK1.A

Example of a text file to define the output format:

```
"TEST RECORD BUCKET WHEEL GRAVITY CENTER";
"----------------------------------------------------------------------------";
"NO. OF MEASURED VALUES : = 1";
"*********************************************";
"X1 = %4.3LF", Q31;
"Y1 = %4.3LF", Q32;
"Z1 = %2LF", Q33;
"*********************************************";
```

Associated file %D16SIM.A

```
TEST  RECORD  BUCKET  WHEEL  GRAVITY  CENTER
----------------------------------------------------------------------------
NO. OF MEASURED VALUES : = 1
*********************************************
X1 = 149.360
Y1 = 25.509
Z1 =   37
*********************************************
```

Special character	Function
"............"	Define output format for texts and variables between the quotation marks
%4.3LF	Define format for variables (long, floating): 4 places before and 3 places behind the decimal point
,	Separation character between output format and parameter
;	End of block character

Reading system data

```
D18
SYS-DATUM
READ
```

With the function D18: SYS-DATUM READ you can read system data and store them in Q parameters. You select the system data through a group number and additionally through a number and an index.

Group name	Group No.	Number	Index	System data
Program information	10	1		MM/inch condition
		2		Overlap factor for pocket milling
		3		Number of active fixed cycle
Machine status	20	1		Active tool number
		2		Prepared tool number
		3		Active tool axis
		4		Programmed spindle rpm
		5		Active spindle status
		8		Coolant status
		9		Active feed rate
Data from the tool table	50	1		Tool length
		2		Tool radius
		3		Tool radius R2
		4		Oversize for tool length DL
		5		Oversize for tool radius DR
		6		Oversize for tool radius DR2
		7		Tool inhibited (0 or 1)
		8		Number of replacement tool
		9		Maximum tool age TIME1
		10		Maximum tool age TIME2
		11		Current tool age CUR.TIME
		12		PLC status
		13		Maximum tooth length LCUTS
		14		Maximum plunge angle ANGLE
		15		TT: Number of teeth CUT.
		16		TT: Wear tolerance for length LTOL
		17		TT: Wear tolerance for radius RTOL
		18		TT: Rotational direction DIRECT (3 or 4)
		19		TT: Offset for radius R-OFFS
		20		TT: Offset for length L-OFFS
		21		TT: Break tolerance in length LBREAK
		22		TT: Break tolerance in radius RBREAK

Group name	Group No.	Number	Index	System data
Active transformations	210	1	–	Basic rotation in MANUAL mode
		2	–	Rotation programmed with Cycle 10
		3	–	Active mirror axis
				0: mirroring not active
				+1: X axis mirrored
				+2: Y axis mirrored
				+4: Z axis mirrored
				+8: IV axis mirrored
				+16: V axis mirrored
				Combinations = sum of individual axes
		4	1	Active scaling factor in X axis
		4	2	Active scaling factor in Y axis
		4	3	Active scaling factor in Z axis
		4	4	Active scaling factor in IV axis
		4	5	Active scaling factor in V axis
		5	1	3D ROT A axis
		5	2	3D ROT B axis
		5	3	3D ROT C axis

Example: Read a BASIC ROTATION from the MANUAL mode
and store it in Q25
D18 Q25 P01 210 P02 1

Transfer to the PLC

```
D19
PLC=
```

The function D19: PLC transfers up to two numerical values or
Q parameter contents to the PLC.

Increments and units: 0.1 µm or 0.0001°

Example D19 P01 +10 P02 +Q3

The numerical value 10 means 1 µm or 0.001°.

7.7 Entering Formulas Directly

You can enter mathematical formulas that include several operations either by soft key or directly from the ASCII keyboard. We recommend entering the operations by soft key, since this eliminates the possibility of syntax errors.

Overview of functions

Mathematical function	Soft key
Addition Example: Q10 = Q1 + Q5	**+**
Subtraction Example: Q25 = Q7 – Q108	**–**
Multiplication Example: Q12 = 5 ∗ Q5	**∗**
Division Example: Q25 = Q1 / Q2	**/**
Open parentheses Example: Q12 = Q1 ∗ **(**Q2 + Q3)	**(**
Close parentheses Example: Q12 = Q1 ∗ (Q2 + Q3**)**	**)**
Square Example: Q15 = SQ 5	**SQ**
Square root Example: Q22 = SQRT 25	**SQRT**
Sine of an angle Example: Q44 = SIN 45	**SIN**
Cosine of an angle Example: Q45 = COS 45	**COS**
Tangent of an angle Example: Q46 = TAN 45	**TAN**

Mathematical function	Soft key
Arc sine: Inverse of the sine. Determine the angle from the ratio of the opposite side to the hypotenuse. Example: Q10 = ASIN 0.75	ASIN
Arc cosine: Inverse of the cosine. Determine the angle from the ratio of the adjacent side to the hypotenuse. Example: Q11 = ACOS Q40	ACOS
Arc tangent: Inverse of the tangent. Determine the angle from the ratio of the opposite to the adjacent side. Example: Q12 = ATAN Q50	ATAN
Powers (x^y) Example: Q15 = 3^3	^
π (3.14159)	PI
Natural logarithm (LN) of a number, base 2.7183 Example: Q15 = LN Q11	LN
Logarithm of a number, base 10 Example: Q33 = LOG 022	LOG
Exponential function (2.7183^n) Example: Q1 = EXP Q12	EXP
Negate (multiply by −1) Example: Q2 = NEG Q1	NEG
Drop places after decimal point (form an integer) Example: Q3 = INT Q42	INT
Absolute value Example: Q4 = ABS Q22	ABS
Drop places before the decimal point (form a fraction) Example: Q5 = FRAC Q23	FRAC

Rules for formulas

- Higher-level operations are performed first (multiplication and division before addition and subtraction):

  $Q1 = 5 \times 3 + 2 \times 10 = 35 \Rightarrow$ 1st step: $5 \times 3 = 15$
  2nd step: $2 \times 10 = 20$
  3rd step: $15 + 20 = 35$

  $Q2 = SQ\ 10 - 3\char94 3 = 73 \quad \Rightarrow$ 1st step: $10^2 = 100$
  2nd step: $3^3 = 27$
  3rd step: $100 - 27 = 73$

- Distributive law:
  $a(b + c) = ab + ac$

Programming example

Calculate an angle with arc tangent as opposite side (Q12) and adjacent side (Q13), then store in Q25.

PARA-METER ▶	FORMULA	Select the formula entering function.
PARAMETER NUMBER FOR RESULT ?		
	2 5 ENT	Enter the parameter number, for example Q25.
▷	ATAN	Shift the soft-key row to the right. Select arc tangent.
◁	(	Shift the soft-key row back to the left. Open parentheses.
	Q 1 2	Enter parameter number Q12.
	/	Select division.
	Q 1 3) END	Enter parameter number Q13. Close parentheses; conclude formula entry.

Resulting NC block: Q25 = ATAN (Q12 / Q13)

7.8 Measuring with the 3D Touch Probe During Program Run

The 3D touch probe can measure positions on the workpiece while the program is being run.

Applications:

- Measuring differences in the height of cast surfaces
- Tolerance checking during machining

To program the use of a touch probe, press the TOUCH PROBE key. You pre-position the probe to automatically probe the desired position. The coordinate measured for the probe point is stored under a Q parameter.

The TNC interrupts the probing process if the stylus is not deflected within a certain distance (selectable via MP6130).

Upon contact, the position coordinates of the probe are stored in the parameters Q115 to Q119. The stylus length and radius are not included in these values.
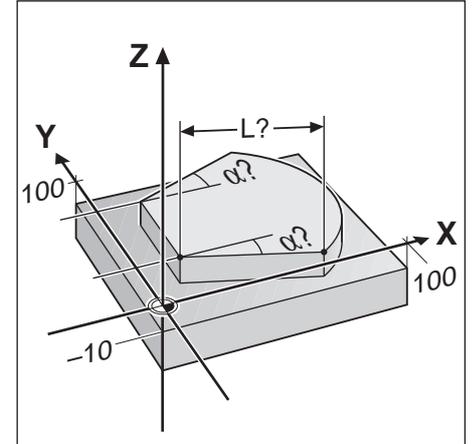
Fig. 7.4:   Dimensions to be measured

- Pre-position the probe manually to avoid a collision when the programmed pre-positioning point is approached.
- Use the tool data (length, radius, axis) either from the calibrated data or from the last TOOL CALL block. Selection is made with machine parameter MP 7411 (see page 11-13).

To program the use of a touch probe:

PARAMETER NUMBER FOR RESULT ?

5 ENT	Enter the number of the Q parameter to which the coordinate should be assigned, for example Q5.

PROBING AXIS/PROBING DIRECTION ?

X **⁻/+** ENT	Enter the probing axis for the coordinate, for example X. Select and confirm the probing direction.

X **5** **Y** **0** **Z** **⁻/+** **5**	Enter all coordinates for the pre-positioning point values, for example X = 5 mm, Y = 0, Z = –5 mm.

END	Conclude input.

*Resulting NC block: G55 P01 Q5 P02 X– X+5 Y+0 Z–5 ***

Example for exercise: Measuring the height of an island on a workpiece

Coordinates for pre-positioning the 3D touch
probe

Touch point 1:	X	=	20	mm	(Q11)
	Y	=	50	mm	(Q12)
	Z	=	10	mm	(Q13)
Touch point 2:	X	=	50	mm	(Q21)
	Y	=	10	mm	(Q22)
	Z	=	0	mm	(Q23)
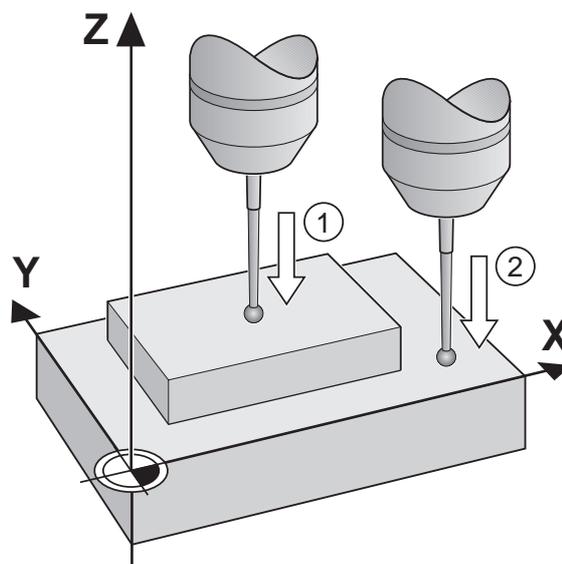
Part program

%S717I G71 * .... Start of program
N10 D00 Q11 P01 +20 *
N20 D00 Q12 P01 +50 *
N30 D00 Q13 P01 +10 * .... Assign coordinates to the parameters for pre-positioning
N40 D00 Q21 P01 +50 * .... the touch probe
N50 D00 Q22 P01 +10 *
N60 D00 Q23 P01 +0 *
N70 T0 G17 *
N80 G00 G40 G90 Z+100 M06 * .... Insert probe
N90 G55 P01 10 P02 Z– X+Q11 Y+Q12 Z+Q13 * ..... Probe in negative direction; store Z coordinate in Q10 (first
                                                                                   point)
N100 X+Q21 Y+Q22 * .... Intermediate positioning for second measurement
N110 G55 P01 20 P02 Z- X+Q21 Y+Q22 Z+Q23 * .... Probe in negative direction; store Z coordinate in Q20 (second
                                                                                   point)
N120 D02 Q1 P01 +Q20 P02 +Q10 * .... Measure height of island and assign to Q1
N130 G38 * .... Program stop; Q1 can be checked (see also page 7-13)
N140 Z+100 M02 * .... Retract in the infeed axis and end the program
N99999 %S717I G71 *

7.9 Programming Examples

Rectangular pocket with island, corner rounding and tangential approach

Pocket center coordinates:	X	=	50 mm (Q1)
	Y	=	50 mm (Q2)
Pocket length	X	=	90 mm (Q3)
Pocket width	Y	=	70 mm (Q4)
Working depth	Z	=	(–)15 mm (–Q5)
Corner radius	R	=	10 mm (Q6)
Milling feed rate	F	=	200 mm/min (Q7)
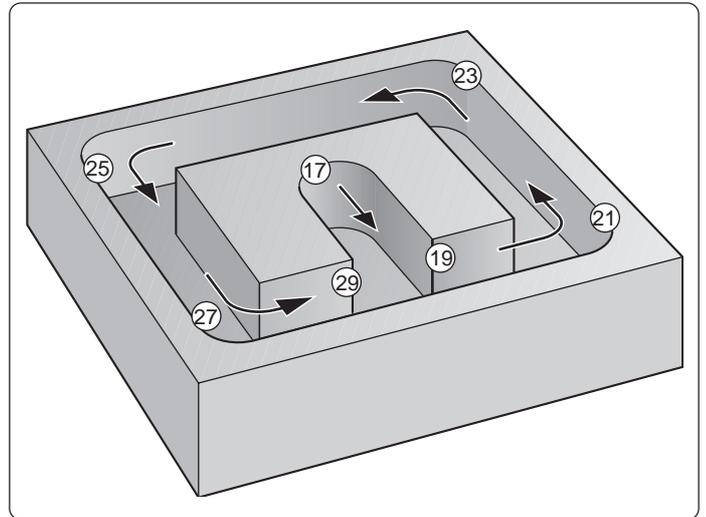
Part program

```
%S77I G71 * ........................................... Start of program
N10 D00 Q1 P01 +50 *
N20 D00 Q2 P01 +50 *
N30 D00 Q3 P01 +90 * ............................... Assign pocket data to the Q parameters
N40 D00 Q4 P01 +70 *
N50 D00 Q5 P01 +15 *
N60 D00 Q6 P01 +10 *
N70 D00 Q7 P01 +200 *
N80 G30 G17 X+0 Y+0 Z–20 * ................................. Define workpiece blank
N90 G31 X+100 Y+100 Z+0 *
N100 G99 T1 L+0 R+5 * ........................................ Define tool
N110 T1 G17 S1000 * ................................. Call tool
N120 G00 G40 G90 Z+100 M06 * ............................ Retract and insert tool
N130 D04 Q13 P01 +Q3 P02 +2 * ............................ The length of the pocket is halved for the path of traverse in
                                                          block N200
N140 D04 Q14 P01 +Q4 P02 +2 * ............................ The width of the pocket is halved for the paths of traverse in
                                                          blocks N220, N300
N150 D04 Q16 P01 +Q6 P02 +4 * ............................ Rounding radius for tangential approach
N160 D04 Q17 P01 +Q7 P02 +2 * ............................ Feed rate at corners is half the feed rate for linear traverse
```

7.9   Programming Examples

N170 X+Q1 Y+Q2 M03 * .... Pre-position in X/Y (pocket center), spindle ON
N180 Z+2 * .... Pre-position over workpiece
N190 G01 Z–Q5 FQ7 * .... Move at feed rate Q7 (= 100) to working depth –Q5
(= –15mm)
N200 G41 G91 X+Q13 G90 Y+Q2 * .... First contour point on the side
N210 G26 RQ16 * .... Soft (tangential) approach
with radius Q16 (= 5 mm)
N220 G91 Y+Q14 *
N230 G25 RQ6 *
N240 X–Q3 *
N250 G25 RQ6 *
N260 Y–Q4 * .... Mill sides of rectangular pocket (incremental)
N270 G25 RQ6 *
N280 X+Q3 *
N290 G25 RQ6 *
N300 Y+Q14 *
N310 G27 RQ16 * .... Soft (tangential) departure
N320 G00 G40 G90 X+Q1 Y+Q2 * .... Depart contour (absolute to pocket center), cancel radius
compensation
N330 Z+100 M02 * .... Retract in the infeed axis
N99999 %S77I G71 *

Bolt hole circles

Bore pattern distributed over a full circle:

The entry values are listed in the program below in blocks N10 to N80.

Movements in the plane are programmed with polar coordinates.

Bore pattern distributed over a circle sector:

The entry values are listed below in blocks N150 to N190; Q5, Q7 and Q8 remain the same.
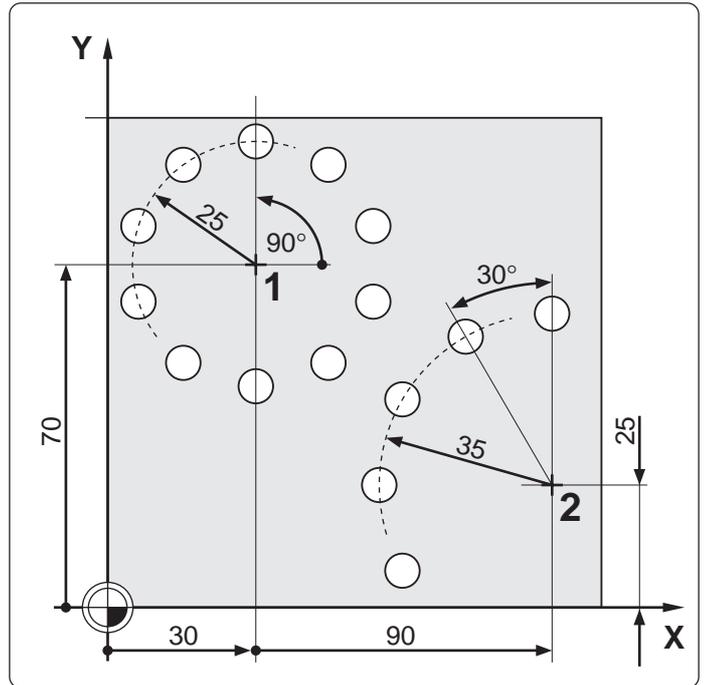
Part program

```
% BOLTCIRC G71 *
N10 D00 Q1 P01 +30 *
N20 D00 Q2 P01 +70 *
N30 D00 Q3 P01 +11 *
N40 D00 Q4 P01 +25 *
N50 D00 Q5 P01 +90 *
N60 D00 Q6 P01 +0 *
N70 D00 Q7 P01 +2 *
N80 D00 Q8 P01 +15 *
N90 G30 G17 X+0 Y+0 Z–20 *
N100 G31 G90 X+100 Y+100 Z+0 *
N110 G99 T1 L+0 R+4 *
N120 T1 G17 S2500 *
N130 G83 P01 +Q7 P02 –Q8 P03 +5
P04 0 P05 250 *
N140 L1,0 *

N150 D00 Q1 P01 +90 *
N160 D00 Q2 P01 +25 *
N170 D00 Q3 P01 5 *
N180 D00 Q4 P01 +35* 
N190 D00 Q6 P01 +30*
N200 L1,0 *
N210 G00 G40 G90 Z+200 M2 *
```

Load data for bolt hole circle 1
Circle center X coordinate
Circle center Y coordinate
Number of holes
Bolt circle radius
Starting angle
Hole angle increment (0: distribute holes over 360°)
Setup clearance
Total hole depth

Cycle definition: Pecking
Call bolt hole circle 1
Load data for bolt hole circle 2 (only re-enter changed data)
New circle center X coordinate
New circle center Y coordinate
New number of holes
New bolt circle radius
New hole angle increment (not full circle, 5 holes 30° apart)
Call bolt hole circle 2

7.9   Programming Examples

```
N220 G98 L1 * ...........................................................Subprogram bolt hole circle
N230 D00 Q10 P01 +0 * ............................................Set the counter for finished holes
N240 D10 P01 +Q6 P02 +0 P03 +10 * .....................If the hole angle increment has been entered, jump to LBL 10
N250 D04 Q6 P01 +360 P02 +Q3 * ...........................Calculate the hole angle increment, distribute holes over 360°
N260 G98 L10 *
N270 D01 Q11 P01 +Q5 P02 +Q6 * .........................Calculate second hole position from the start angle and hole
                                                         angle increment
N280 G90 I+Q1 J+Q2 G00 G40 * ..............................Set pole at bolt circle center
N290 G10 R+Q4 H+Q5 M3 * ...................................Move in the plane to first hole
N300 G00 Z+Q7 M99 * .............................................Move in Z to setup clearance, call cycle
N310 D01 Q10 P01 +Q10 P02 +1 * .........................Count completed holes
N320 D09 P01 +Q10 P02 +Q3 P03 99 * .................Finished?
N330 G98 L2 *
N340 G10 G40 G90 R+Q4 H+Q11 M99 * ...............Drill second hole and further holes
N350 D01 Q10 P01 +Q10 P02 +1 * .........................Count finished holes
N360 D01 Q11 P01 +Q11 P02 +Q6 * ......................Calculate angle for next hole
N370 D12 P01 +Q10 P02 +Q3 P03 2 * ...................Not finished?
N380 G98 L99 *
N390 G00 G91 Z+200 * ...........................................Retract in Z
N400 G98 L0 * .......................................................End of subprogram
N99999 % BOLTCIRC G71 *
```

Ellipse

X-coordinate calculation: X = *a* cos α
Y-coordinate calculation: Y = *b* sin α

a, b : Semimajor and semiminor axes of
the ellipse
α : Angle between the leading axis and
the connecting line from P to the
center of the ellipse.
X_0, Y_0: Center of the ellipse

The points of the ellipse are calculated and
connected by many short lines. The more
points that are calculated and the shorter the
lines connecting them, the smoother the
curve becomes.

The machining direction can be altered by
changing the entries for the starting angle
and end angle.

The input parameters are listed below in
blocks N10 to N120. Calculations are
programmed with the FORMULA function.
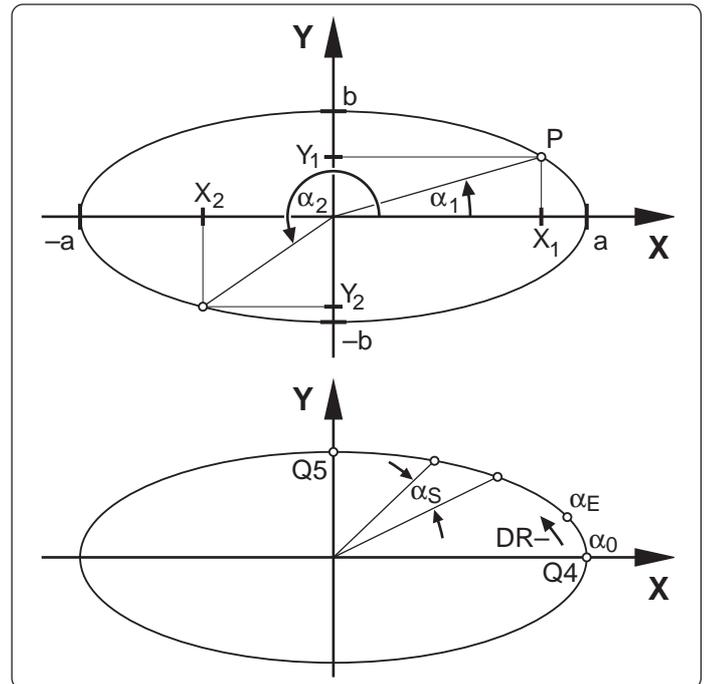
Part program

```
% Ellipse G71 * ..........................................................Load data
N10 D00 Q1 P01 +50 * ...............................................X coordinate for center of ellipse
N20 D00 Q2 P01 +50 * ...............................................Y coordinate for center of ellipse
N30 D00 Q3 P01 +50 * ...............................................Semiaxis in X
N40 D00 Q4 P01 +20 * ...............................................Semiaxis in Y
N50 D00 Q5 P01 +0 * .................................................Starting angle
N60 D00 Q6 P01 +360 * .............................................End angle
N70 D00 Q7 P01 +40 * ...............................................Number of calculation steps
N80 D00 Q8 P01 +0 * .................................................Rotational position
N90 D00 Q9 P01 +10 * ...............................................Depth
N100 D00 Q10 P01 +100 * ........................................Plunging feed rate
N110 D00 Q11 P01 +350 * ........................................Milling feed rate
N120 D00 Q12 P01 +2 * ............................................Setup clearance in Z
N130 G30 G17 X+0 Y+0 Z-20 *
N140 G31 G90 X+100 Y+100 Z+0 *
N150 G99 T1 L+0 R+2.5 *
N160 T1 G17 *
N170 G00 G40 G90 Z+200 *
N180 L10,0 * ............................................................Execute subprogram ellipse
N190 G00 Z+200 M2*
```

Continued on next page...

7.9   Programming Examples

```
N200 G98 L10 *
N210 G54 X+Q1 Y+Q2 * ......................................... Shift datum to center of ellipse
N220 G73 G90 H+Q8 * .......................................... Activate rotation if Q8 is loaded
N230 Q35 = (Q6-Q5)/Q7 ....................................... Calculate angle increment (end angle to starting angle
                                                             divided by the number of steps)
N240 Q36 = Q5 ................................................... Set current angle for calculation = starting angle
N250 Q37 = 0 ..................................................... Set counter for milled steps
N260 Q21 = Q3 * COS Q36 ................................... Calculate X coordinate for starting point
N270 Q22 = Q4 * SIN Q36 .................................... Calculate Y coordinate for starting point
N280 G00 G40 G90 X+Q21 Y+Q22 M3 * ................ Move to starting point in the plane
N290 Z+Q12 * ................................................... Rapid traverse in Z to setup clearance
N300 G01 Z-Q9 FQ10 * ......................................... Plunge to milling depth at plunging feed rate
N310 G98 L1 *
N320 Q36 = Q36 + Q35 ......................................... Update the angle
N330 Q37 = Q37 + 1 ............................................ Update the counter
N340 Q21 = Q3 * COS Q36 ................................... Calculate the next X coordinate
N350 Q22 = Q4 * SIN 036 ..................................... Calculate the next Y coordinate
N360 G01 X+Q21 Y+Q22 FQ11 .............................. Move to next point
N370 D12 P01+Q37 P02+Q7 P031 * ....................... Not finished?
N380 G73 G90 H+0 * ........................................... Reset rotation
N390 G54 * ........................................................ Reset datum shift
N400 G00 G40 G90 Z+Q12 * ................................. Move in Z to setup clearance
N410 G98 L0 * .................................................... End of subprogram
N99999 % ELLIPSE G71 *
```

Hemisphere machined with end mill

Notes on the program:

- The tool moves upward in the Z/X plane.
- You can enter an oversize in block 12 (Q12) if you want to machine the contour in several steps.
- The tool radius is automatically compensated with parameter Q108.

The program works with the following quantities:

- Solid angle: Starting angle   Q1
                    End angle   Q2
                    Increment   Q3
- Sphere radius   Q4
- Setup clearance   Q5
- Plane angle: Starting angle   Q6
                    End angle   Q7
                    Increment   Q8
- Center of sphere: X coordinate   Q9
                    Y coordinate   Q10
- Milling feed rate   Q11
- Oversize   Q12

The parameters additionally defined in the program have the following meanings:

- Q15:   Setup clearance above the sphere
- Q21:   Solid angle during machining
- Q24:   Distance from center of sphere to tool center
- Q26:   Plane angle during machining
- Q108:   TNC parameter with tool radius
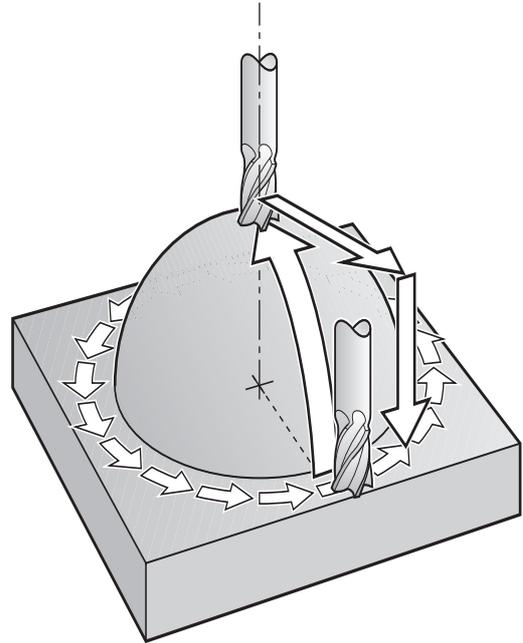
Part program

```
%S712I G71 * .......................................... Start of program
N10 D00 Q1 P01 +90 *
N20 D00 Q2 P01 +0 *
N30 D00 Q3 P01 +5 *
N40 D00 Q4 P01 +45 *
N50 D00 Q5 P01 +2 *
N60 D00 Q6 P01 +0 *
N70 D00 Q7 P01 +360 *
N80 D00 Q8 P01 +5 *
N90 D00 Q9 P01 +50 *
N100 D00 Q10 P01 +50 *
N110 D00 Q11 P01 +500 *
N120 D00 Q12 P01 +0 * ........................... Assign the sphere data to the parameters
N130 G30 G17 X+0 Y+0 Z–50 * ................... Define workpiece blank
N140 G31 G90 X+100 Y+100 Z+0 *
N150 G99 T1 L+0 R+5 * ........................... Define tool
N160 T1 G17 S2500 * ............................... Call tool
N170 G00 G40 G90 Z+100 M06 * ............. Retract and insert tool
N180 L10,0 * ........................................... Call subprogram
N190 Z+100 M02 * ................................... Retract in the infeed axis; return to beginning of program
```

Continued on next page...

```
N200 G98 L10 *
N210 D01 Q15 P01 +Q5 P02 +Q4 *
N220 D00 Q21 P01 +Q1 * ........................................ Determine starting and calculation values
N230 D01 Q24 P01 +Q4 P02 +Q108 *
N240 D00 Q26 P01 +Q6 *
N250 G54 X+Q9 Y+Q10 Z–Q4 * ............................... Shift datum to center of sphere
N260 G73 G90 H+Q6 * ............................................ Rotation for program start (starting plane angle)
N270 I+0 J+0 *
N280 G11 R+Q24 H+Q6 FQ11 * ............................... Pre-positioning before machining
N290 G98 L1 *
N300 I+Q108 K+0 * ................................................ Set pole (X/Z plane)
N310 G01 Y+0 Z+0 FQ11 * ....................................... Pre-positioning at each arc beginning
N320 G98 L2 *
N330 G11 R+Q4 H+Q21 FQ11 *
N340 D02 Q21 P01 +Q21 P02 +Q3 * ........................ Mill the sphere upward until the highest point is reached
N350 D11 P01 +Q21 P02 +Q2 P03 2 *
N360 G11 R+Q4 H+Q2 * .......................................... Mill the highest point on the sphere
N370 G00 Z+Q15 * .................................................. Retract in Z
N380 X+Q24 * ......................................................... Retract in X
N390 D01 Q26 P01 +Q26 P02 +Q8 * ........................ Prepare the next rotation increment
N400 D00 Q21 P01 +Q1 * ........................................ Reset solid angle for machining to the starting value
N410 G73 G90 H+Q26 * ........................................... Activate rotation for next operation
N420 D12 P01 +Q26 P02 +Q7 P03 1 *
N430 D09 P01 +Q26 P02 +Q7 P03 1 * ..................... Rotate the coordinate system around the Z axis until the end
                                                                                          plane angle is reached
N440 G73 G90 H+0 * ............................................... Reset rotation
N450 G54 X+0 Y+0 Z+0 * ........................................ Reset data shift
N460 G98 L0 * ........................................................ End of subprogram
N99999 %S712I G71 *
```

8.1 General Overview of Cycles

Frequently recurring machining sequences that comprise several working steps are stored in the control memory as standard cycles. Coordinate transformations and other special functions are also provided as standard cycles.

These cycles are grouped into the following types:

- **Simple fixed cycles** such as pecking and tapping, as well as the milling operations slot milling, rectangular pocket milling and circular pocket milling.
- **SL (Subcontour List) Cycles**, group I. These allow machining of relatively complex contours composed of several overlapping subcontours.
- **SL Cycles**, group II, for contour-oriented machining. During rough-out and finishing, the tool follows the contour as defined in the SL cycles. The cutter infeed positions are determined automatically by the control.
- **Coordinate transformation cycles**. These enable datum shifts, rotation, mirroring, enlarging and reducing for various contours.
- **Special cycles** such as dwell time, program call, and oriented spindle stop.

Programming a cycle

Defining a cycle

Enter the G function for the desired cycle and program it in the dialog. The following example illustrates how cycles are defined:

G **8** **5** ENT	Select a cycle, such as Rigid Tapping.
SETUP CLEARANCE ?	
⁻/+ **2** ENT	Enter the setup clearance (here, –2 mm).
TOTAL HOLE DEPTH ?	
⁻/+ **3** **0** ENT	Enter the total hole depth (here, –30 mm).
THREAD PITCH ?	
0 **·** **7** **5** END	Enter the thread pitch (here, +0.75 mm).

Resulting NC block: G85 P01 –2 P02 –30 P03 +0.75 *

Cycle call

The following cycles become effective automatically as soon as they are defined in the part program:

• Coordinate transformation cycles
• Dwell time cycle
• SL cycles which determine the contour and the global parameters

All other cycles must be called separately. Further information on cycle calls is provided in the descriptions of the individual cycles.

If the cycle is to be executed after the block in which it was called, program the cycle call

• with G79
• with miscellaneous function M99.

If the cycle is to be executed after every positioning block, it must be called with miscellaneous function M89 (depending on the machine parameters).

M89 is cancelled with   • M99
                            • G79
                            • A new cycle definition

Prerequisites:

The following data must be programmed before a cycle call:

• Blank form for graphic display
• Tool call
• Positioning block for starting position X, Y with tool radius compensation G40
• Positioning block for starting position Z (setup clearance)
• Direction of spindle rotation (miscellaneous functions M3/M4)
• Cycle definition

Dimensions in the tool axis

The dimensions for the tool axis are always referenced to the position of the tool at the time of the cycle call, and are interpreted by the control as incremental dimensions. It is not necessary to program G91.

The control assumes that the tool is located at clearance height over the workpiece at the beginning of the cycle (except for SL cycles of group II).

8.2 Simple Fixed Cycles

PECKING (G83)

Process

- The tool drills from the starting point to the first pecking depth at the programmed feed rate.
- When it reaches the first pecking depth, the tool retracts in rapid traverse to the starting position and advances again to the first pecking depth minus the advanced stop distance *t* (see calculations).
- The tool advances with another infeed at the programmed feed rate.
- Drilling and retracting are performed alternately until the programmed total hole depth is reached.
- After the dwell time at the hole bottom, the tool is retracted to the starting position in rapid traverse for chip breaking.
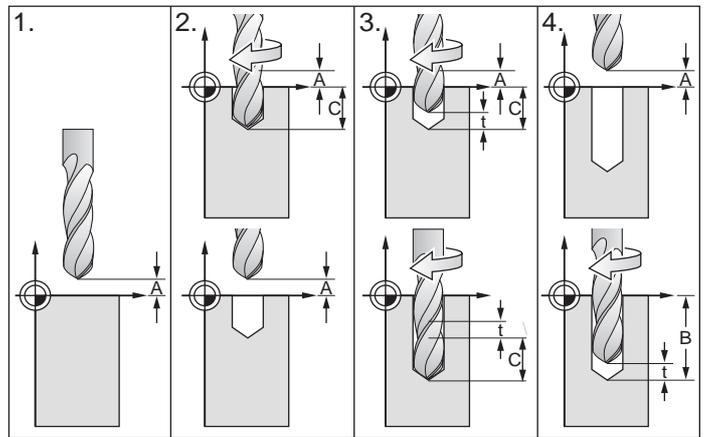
Fig. 8.1: PECKING cycle

Input data

- SETUP CLEARANCE Ⓐ:
  Distance between tool tip (at starting position) and workpiece surface
- TOTAL HOLE DEPTH Ⓑ:
  Distance between workpiece surface and bottom of hole (tip of drill taper). The algebraic sign determines the working direction (a negative value means negative working direction).
- PECKING DEPTH Ⓒ:
  Infeed per cut.
  If the TOTAL HOLE DEPTH equals the PECKING DEPTH, the tool will drill to the programmed total hole depth in one operation.
  The PECKING DEPTH does not have to be a multiple of the TOTAL HOLE DEPTH.
  If the PECKING DEPTH is programmed greater than the TOTAL HOLE DEPTH, the tool only advances to the specified TOTAL HOLE DEPTH.
- DWELL TIME in seconds:
  Amount of time the tool remains at the total hole depth for chip breaking.
- FEED RATE:
  Traversing speed of the tool during drilling.

Calculations

The advanced stop distance *t* is automatically calculated by the control:

- At a total hole depth of up to 30 mm, t = 0.6 mm
- At a total hole depth exceeding 30 mm, t = total hole depth / 50
  Maximum advanced stop distance: 7 mm

Example: PECKING

Hole coordinates:

①   X = 20 mm   Y = 30 mm

②   X = 80 mm   Y = 50 mm

Hole diameter: 6   mm

Setup clearance: 2   mm

Total hole depth: 15   mm

Pecking depth: 10   mm

Dwell time: 1   s

Feed rate: 80   mm/min
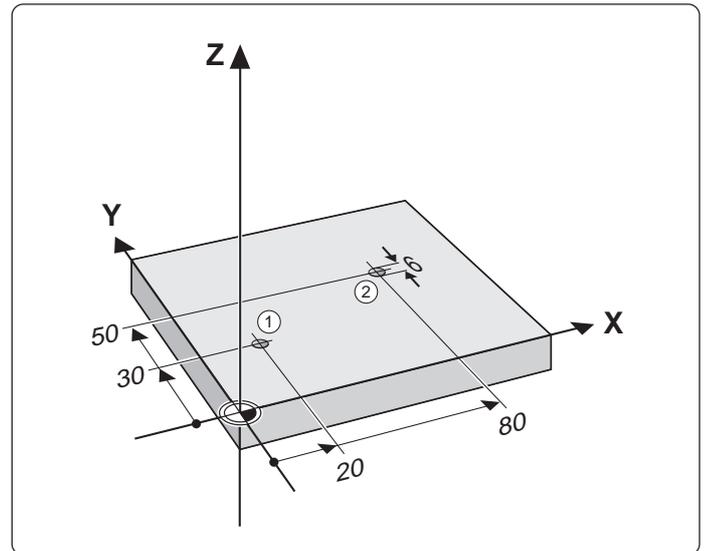
PECKING cycle in a part program

```
%S85I G71 * ............................................................ Start of program
N10 G30 G17 X+0 Y+0 Z–20 * ................................. Define workpiece blank
N20 G31 G90 X+100 Y+100 Z+0 *
N30 G99 T1 L+0 R+3 * ............................................ Define tool
N40 T1 G17 S1200 * .................................................. Call tool
N50 G83 P01 –2 P02 –15 P03 –10 P04 1 P05 80 * .... Define PECKING cycle
N60 G00 G40 G90 Z+100 M06 * .............................. Retract in the infeed axis, insert tool
N70 X+20 Y+30 M03 * ............................................. Pre-position for the first hole, spindle ON
N80 Z+2 M99 * ......................................................... Pre-position in Z to setup clearance, call cycle
N90 X+80 Y+50 M99 * ............................................. Move to second hole, call cycle
N100 Z+100 M02 * ................................................... Retract in the infeed axis, end of program
N99999 %S85I G71 *
```

TAPPING with floating tap holder (G84)

Process

- The thread is cut in one pass.
- Once the tool has reached the total hole depth, the direction of spindle rotation is reversed and the tool is retracted to the starting position at the end of the dwell time.
- At the starting position, the direction of spindle rotation reverses once again.

Required tool

A floating tap holder is required. It must compensate the tolerances between feed rate and spindle speed during the tapping process.
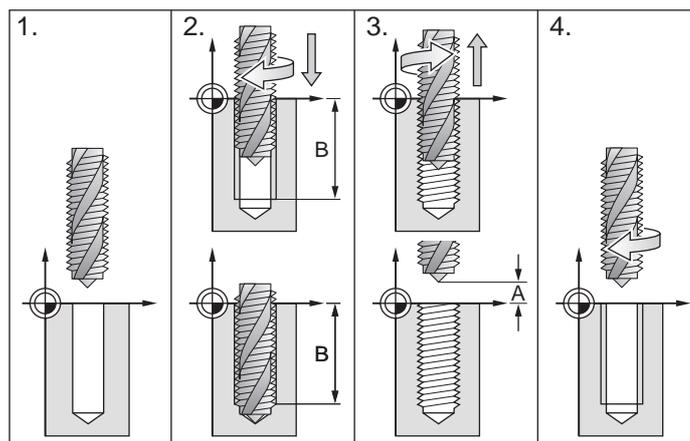
Fig. 8.2: TAPPING cycle

Input data

- SETUP CLEARANCE Ⓐ:
  Distance between tool tip (at starting position) and workpiece surface.
  Standard value: approx. 4 x thread pitch
- TOTAL HOLE DEPTH Ⓑ (thread length):
  Distance between workpiece surface and end of thread. The algebraic sign determines the working direction (a negative sign means negative working direction).
- DWELL TIME:
  Enter a dwell time between 0 and 0.5 seconds to avoid wedging of the tool during retraction (further information is available from the machine manufacturer).
- FEED RATE:
  Traversing speed of the tool during tapping.

Calculations

The feed rate is calculated as follows:

$$F = S \times p$$

where F is the feed rate (mm/min), S is the spindle speed (rpm) and p is the thread pitch (mm).

- When a cycle is being run, the spindle speed override knob is disabled. The feed rate override knob is only active within a limited range (preset by the machine manufacturer).
- For tapping right-hand threads activate the spindle with M3, for left-hand threads use M4.

Example: Tapping with a floating tap holder

Cutting an M6 thread at 100 rpm

Tapping coordinates:

X = 50 mm   Y = 20 mm

Pitch   p =   1 mm

$F = S \times p \Rightarrow F = 100 \cdot 1 = 100$ mm/min

Setup clearance: 3   mm

Thread depth: 20   mm

Dwell time: 0.4 s

Feed rate: 100   mm/min
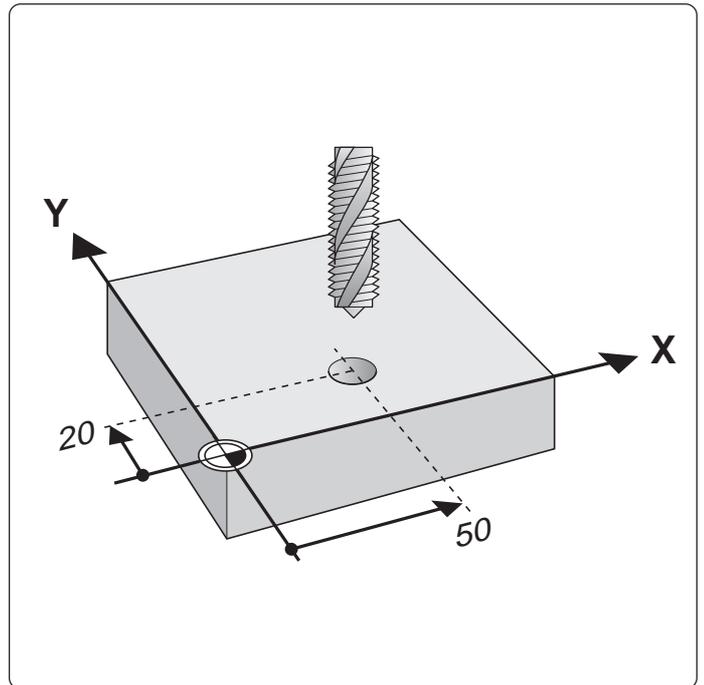
TAPPING cycle in a part program

```
%S87I G71 * ............................................................ Start of program
N10 G30 G17 X+0 Y+0 Z–20 * ................................. Define workpiece blank
N20 G31 G90 X+100 Y+100 Z+0 *
N30 G99 T1 L+0 R+3 * ............................................ Define tool
N40 T1 G17 S100 * .................................................. Call tool
N50 G84 P01 –5 P02 –20 P03 0.4 P04 100 * .......... Define TAPPING cycle
N60 G00 G40 G90 Z+100 M06 * ............................. Retract in the infeed axis, insert tool
N70 X+50 Y+20 M03 * .............................................. Pre-position in the plane, spindle ON
N80 Z+3 M99 * ......................................................... Pre-position in Z to setup clearance, call cycle
N90 Z+100 M02 * ..................................................... Retract in the infeed axis, end of program
N99999 %S87I G71 *
```

RIGID TAPPING (G85)

Machine and control must be specially prepared by the machine manufacturer to enable rigid tapping.

Process

The thread is cut without a floating tap holder in one or several passes.

Rigid tapping offers the following advantages over tapping with a floating tap holder:

- Higher machining speeds possible
- Repeated tapping of the same thread; repetitions are enabled via spindle orientation to the 0° position during cycle call (depending on machine parameter 7160; see page 11-13).
- Increased traverse range of the spindle axis due to absence of a floating tap holder

- The control calculates the feed rate from the spindle speed. If the spindle speed override is used during tapping, the feed rate is automatically adjusted.
- The feed rate override knob is disabled.

Input data

- SETUP CLEARANCE Ⓐ:
  Distance between tool tip (at starting position) and workpiece surface.
- TAPPING DEPTH Ⓑ:
  Distance between workpiece surface (beginning of thread) and end of thread. The algebraic sign determines the working direction: a negative value means negative working direction.
- THREAD PITCH Ⓒ:
  The sign differentiates between right-hand and left-hand threads:
  + = right-hand thread
  − = left-hand thread
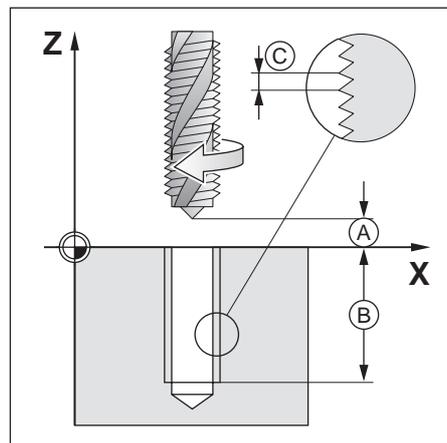
Fig. 8.3:   Input data for RIGID TAPPING cycle

THREAD CUTTING (G86)

Machine and control must be specially prepared by the machine manufacturer to enable thread cutting.

Process

Cycle G86 THREAD CUTTING is performed by means of spindle control. The tool moves with the active spindle speed from its current position to the entered depth. As soon as it reaches the end of thread, spindle rotation is stopped. Tool approach and departure must be programmed separately. The most convenient way to do this is by using OEM cycles. The machine manufacturer can give you further information.

Input data

- THREADING DEPTH (A):
  Distance between current tool position and end of thread. The algebraic sign determines the working direction: a negative value means negative working direction.
- THREAD PITCH (B):
  The sign differentiates between right-hand and left-hand threads:
    + = right-hand thread   (M3 with negative THREADING DEPTH)
    – = left-hand thread   (M4 with negative THREADING DEPTH)
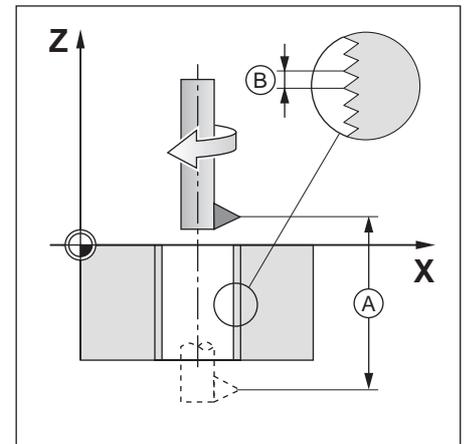
Fig. 8.4:   Input data for THREAD CUTTING cycle

- The control calculates the feed rate from the spindle speed. If the spindle speed override is used during thread cutting, the feed rate is automatically adjusted.
- The feed rate override knob is disabled.
- The TNC automatically activates and deactivates spindle rotation. Do not program M3/M4 before cycle call.

Example: Thread cutting with a threading tool

Cutting M12 threads into through holes in an upward movement

Coordinates of the threaded holes:

X = 20   mm   Y = 20   mm
X = 70   mm   Y = 70   mm

Workpiece thickness: 20   mm

Thread pitch p: 1.75   mm
Spindle speed: 100   rpm

Setup clearance at top: 5   mm
Setup clearance at bottom: 5   mm
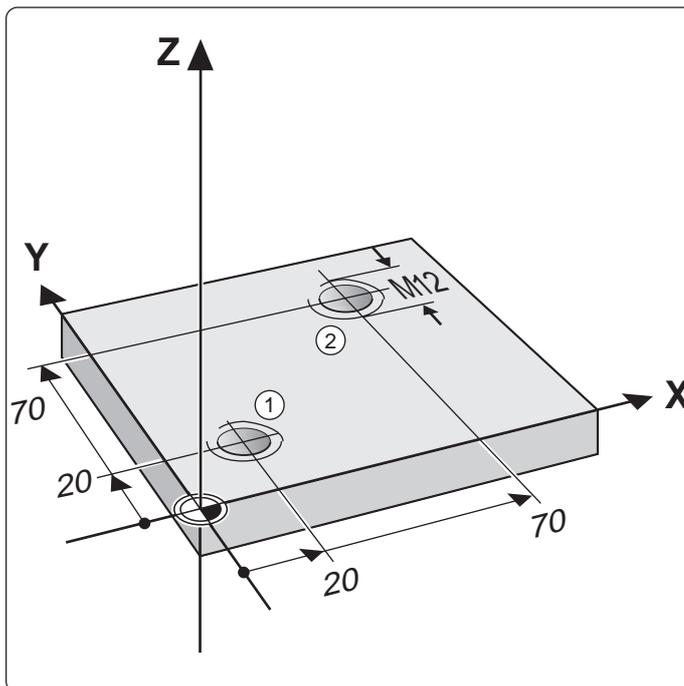
THREAD CUTTING cycle in a part program

```
%C18 G71 *
N10   G30 G17 X+0 Y+0 Z–20*
N20   G31 G90 X+100 Y+100 Z+0*
N30   G99 T1 L+0 R+6*
N40   T1 G17 S100*
N50   G00 G40 G90 Z+50*
N60   G86 P01 +30 P02 –1.75* ................................Threading depth 30 mm, positive direction; thread pitch   1.75
                                                            mm, negative because of upward working direction
N70   X+20 Y+20* ......................................................Approach 1st hole in the X/Y plane
N80   L1,0* ...............................................................Call  subprogram
N90   X+70 Y+70* ......................................................Approach 2nd hole in the X/Y plane
N100 L1,0* ...............................................................Call  subprogram
N110 G00 Z+100 M2* ...............................................End of main program

N120 G98 L1*
N130 G36 S0* ..........................................................Orient spindle to 0° (makes it possible to cut repeatedly )
N140 G00 G40 G91 X–2* .........................................Tool offset in the plane to prevent collision during tool infeed
                                                            (dependent on core diameter)
N150 G00 G90 Z+5* .................................................Pre-position in the tool axis at rapid traverse to setup
                                                            clearance above workpiece surface
N160 G01 Z–30 F 1000* .........................................Pre-position in the tool axis at rapid traverse to bottom
                                                            starting position
N170 G91 X+2* ........................................................Reset the tool in the plane to hole center
N180 G79* ...............................................................Cycle call
N190 G98 L0* ..........................................................End of subprogram
N99999 %C18 G71*
```

SLOT MILLING (G74)

Process

Roughing process:

- The tool penetrates the workpiece from the starting position, offset by the oversize, then mills in the longitudinal direction of the slot.
- The oversize is calculated as: (slot width – tool diameter) / 2.
- After downfeed at the end of the slot, milling is performed in the opposite direction.
  This process is repeated until the programmed milling depth is reached.

Finishing process:

- The control advances the tool at the bottom of the slot on a tangential arc to the outside contour. The tool subsequently climb mills the contour (with M3).
- At the end of the cycle, the tool is retracted in rapid traverse to the setup clearance.
  If the number of infeeds was odd, the tool returns to the starting position at the level of the setup clearance in the main plane.

Fig. 8.5:   SLOT MILLING cycle

Required tool

This cycle requires a center-cut end mill (ISO 1641). The cutter diameter must be smaller than the slot width and larger than half the slot width. The slot must be parallel to an axis of the current coordinate system.

Input data

- SETUP CLEARANCE Ⓐ
- MILLING DEPTH Ⓑ: Slot depth. The algebraic sign determines the working direction (a negative value means negative working direction).
- PECKING DEPTH Ⓒ
- FEED RATE FOR PECKING:
  Traversing speed of the tool during penetration
- FIRST SIDE LENGTH Ⓓ:
  Slot length, specify the sign to determine the first milling direction
- SECOND SIDE LENGTH Ⓔ:
  Slot width
- FEED RATE:
  Traversing speed of the tool in the machining plane.

Fig. 8.6:   Infeeds and distances for the SLOT MILLING cycle

Starting point

Before a cycle is called, the tool must be moved to the following starting point with **tool radius compensation G40:**

- In the tool axis, to setup clearance above the workpiece surface.
- In the machining plane, to the center of the slot (second side length) and, within the slot, offset by the tool radius.
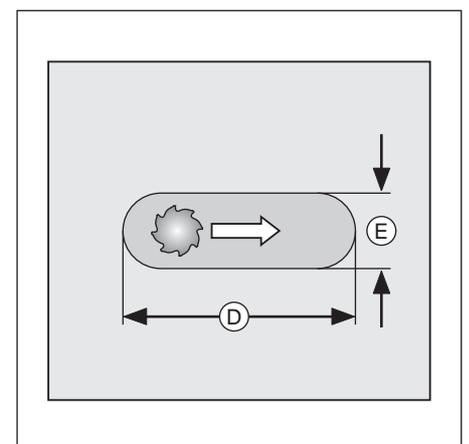
Fig. 8.7:   Side lengths of the slot

Example: Slot milling

A horizontal slot (50 mm x 10 mm) and a vertical slot (80 mm x 10 mm) are to be milled.

The tool radius in the length direction of the slot is taken into account for the starting position.

Starting position, slot ①:
X = 76 mm   Y = 15 mm

Starting position, slot ②:
X = 20 mm   Y = 14 mm

SLOT DEPTH:	15 mm
Setup clearance:	2 mm
Milling depth:	15 mm
Pecking depth:	5 mm
Feed rate for pecking:	80 mm/min

	①	②
Slot length	50 mm	80 mm
1st milling direction	–	+
Slot width:	10 mm	
Feed rate:	120 mm/min	
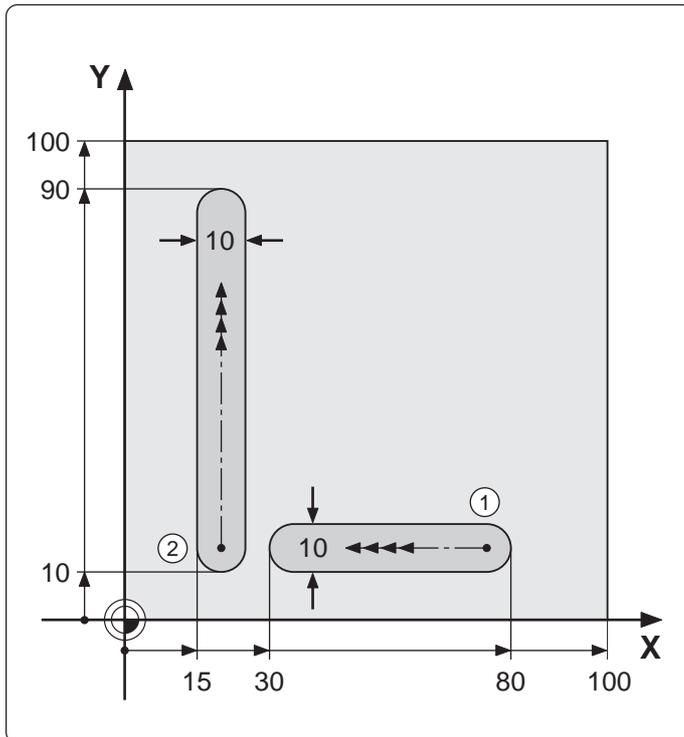
SLOT MILLING cycle in a part program

```
%S810I G71 * ......................................................... Start of program
N10 G30 G17 X+0 Y+0 Z–20 * ................................... Define workpiece blank
N20 G31 G90 X+100 Y+100 Z+0 *
N30 G99 T1 L+0 R+4 * ............................................. Define tool
N40 T1 G17 S2000 * ................................................ Call tool
N50 G74 P01 –2 P02 –15 P03 –5 P04 80 P05 X–50
P06 Y+10 P07 120 * .................................................. Define slot parallel to X axis
N60 G00 G40 G90 Z+100 M06 * .............................. Retract in the infeed axis, insert tool
N70 X+76 Y+15 M03 * ............................................. Approach starting position, spindle ON
N80 Z+2 M99 * ........................................................ Pre-position in Z to setup clearance, cycle call ①
N90 G74 P01 –2 P02 –15 P03 –5 P04 80 P05 Y+80
P06 X+10 P07 120 * .................................................. Define slot parallel to Y axis
N100 X+20 Y+14 M99 * ........................................... Approach starting position, cycle call ②
N110 Z+100 M02 * .................................................. Retract in the infeed axis, end of program
N99999 %S810I G71 *
```

POCKET MILLING (G75/G76)

Process

The rectangular pocket milling cycle is a roughing cycle, in which

- the tool penetrates the workpiece at the starting position (pocket center)
- the tool subsequently follows the programmed path at the specified feed rate (see figure 8.10)

The cutter begins milling in the positive direction of the axis of the longer side. The cutter always starts in the positive Y direction on square pockets. At the end of the cycle, the tool is retracted to the starting position.

Required tool / limitations

The cycle requires a center-cut end mill (ISO 1641) or pilot drilling at the pocket center. The pocket sides are parallel to the axes of the coordinate system.

Direction of rotation for roughing-out

Clockwise: G75
Counterclockwise: G76

Input data

- SETUP CLEARANCE Ⓐ
- MILLING DEPTH Ⓑ
  The algebraic sign determines the working direction (a negative value means negative working direction).
- PECKING DEPTH Ⓒ
- FEED RATE FOR PECKING:
  Traversing speed of the tool during penetration.
- FIRST SIDE LENGTH Ⓓ:
  Pocket length, parallel to the first main axis of the machining plane.
- SECOND SIDE LENGTH Ⓔ:
  Pocket width
  The signs of the side lengths are always positive.
- FEED RATE:
  Traversing speed of the tool in the machining plane.

Calculations

The stepover factor *k* is calculated as follows:

$$k = K \times R$$

where *K* is the overlap factor (preset by the machine manufacturer) and *R* is the cutter radius.

Corner radius

The corner radius is determined by the radius of the milling tool.

Starting point

Before a cycle is called, the tool must be moved to the following starting point with **tool radius compensation G40:**

- In the tool axis, to setup clearance above the workpiece surface.
- In the machining plane, to the pocket center.
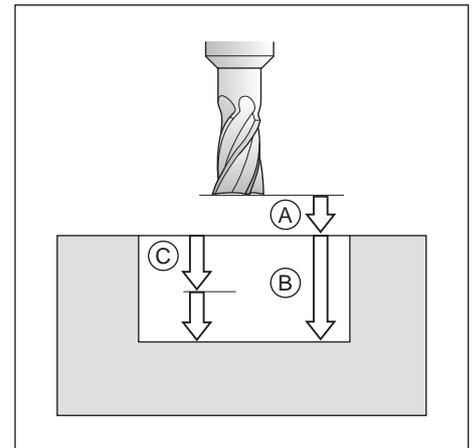
Fig. 8.8:   Infeeds and distances for the POCKET MILLING cycle
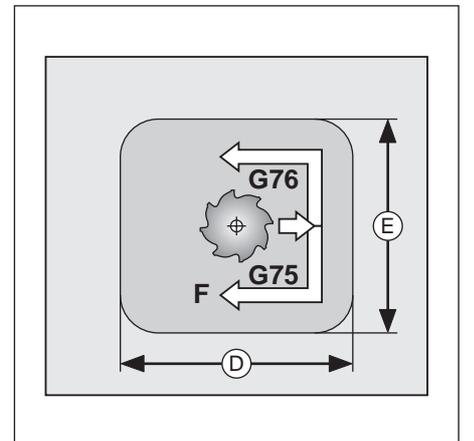
Fig. 8.9:   Side lengths of the pocket
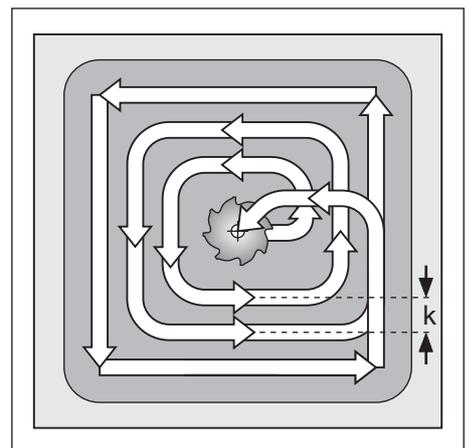
Fig. 8.10:   Tool path for roughing-out

Example: Rectangular pocket milling

Pocket center coordinates:

X = 60   mm   Y = 35   mm

Setup clearance: 2   mm

Milling depth: 10   mm

Pecking depth: 4   mm

Feed rate for pecking: 80   mm/min

First side length: 80   mm

Second side length: 40   mm

Milling feed rate: 100   mm/min

Direction of cutter path: +
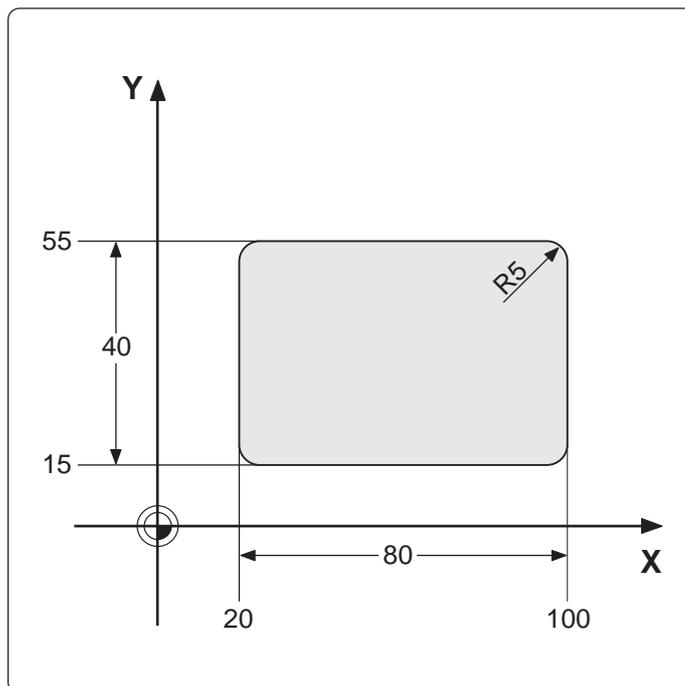
POCKET MILLING cycle in a part program

```
%S812I G71 * ......................................................... Start of program
N10 G30 G17 X+0 Y+0 Z–20 * .................................. Define workpiece blank
N20 G31 G90 X+110 Y+100 Z+0 *
N30 G99 T1 L+0 R+5 * ............................................ Define tool
N40 T1 G17 S2000 * ................................................ Call tool
N50 G76 P01 –2 P02 –10 P03 –4 P04 80 P05 X+80
P06 Y+40 P07 100 * ................................................. Define POCKET MILLING cycle
N60 G00 G40 G90 Z+100 M06 * .............................. Retract in the infeed axis, insert tool
N70 X+60 Y+35 M03 * ............................................. Approach the starting position (center of pocket), spindle ON
N80 Z+2 M99 * ......................................................... Pre-position in Z to setup clearance, cycle call
N90 Z+100 M02 * ..................................................... Retract in the infeed axis, end of program
N99999 %S812I G71 *
```

CIRCULAR POCKET MILLING (G77/G78)

Process

- Circular pocket milling is a roughing cycle in which the tool penetrates the workpiece from the starting position (pocket center).
- The cutter subsequently follows a spiral path (shown in figure 8.11) at the programmed feed rate. The stepover factor is determined by the value *k* (see G75/G76 POCKET MILLING, Calculations).
- The process is repeated until the programmed milling depth is reached.
- At the end of the cycle, the tool is retracted to the starting position.

Required tool

The cycle requires a center-cut end mill (ISO 1641), or pilot drilling at the pocket center.

Direction of rotation for roughing-out

Clockwise: G77
Counterclockwise: G78

Input data

- SETUP CLEARANCE Ⓐ
- MILLING DEPTH Ⓑ: pocket DEPTH.
  The algebraic sign determines the working direction (a negative sign means negative working direction).
- PECKING DEPTH Ⓒ
- FEED RATE FOR PECKING:
  Traversing speed of the tool during penetration
- CIRCLE RADIUS Ⓡ:
  Radius of the circular pocket
- FEED RATE:
  Traversing speed of the tool in the machining plane

Starting point

Before a cycle is called, the tool must be moved to the following starting point with **tool radius compensation G40:**

- In the tool axis, to setup clearance above the workpiece surface.
- In the machining plane, to the pocket center.
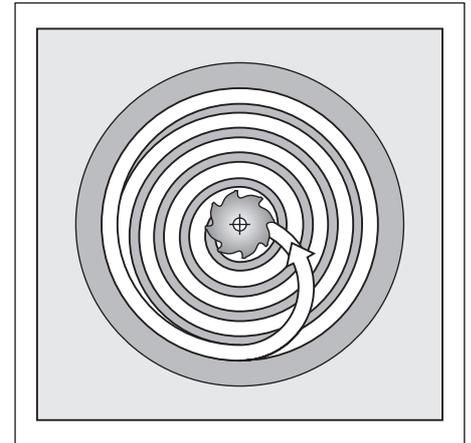
Fig. 8.11:   Cutter path for roughing-out
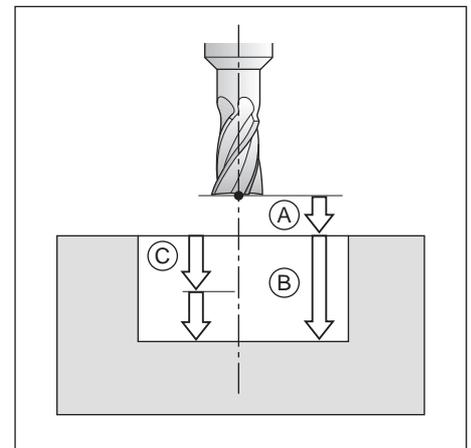
Fig. 8.12:   Distances and infeeds for
              CIRCULAR POCKET MILLING
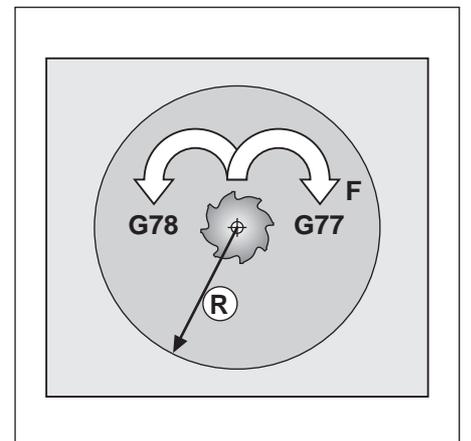
Fig. 8.13:   Direction of the cutter path

Example: Milling a circular pocket

Pocket center coordinates:

X = 60 mm   Y = 50 mm

Setup clearance:	2	mm
Milling depth:	12	mm
Pecking depth:	6	mm
Feed rate for pecking:	80	mm/min
Circle radius:	35	mm
Milling feed rate:	100	mm/min
Direction of the cutter path:	–	
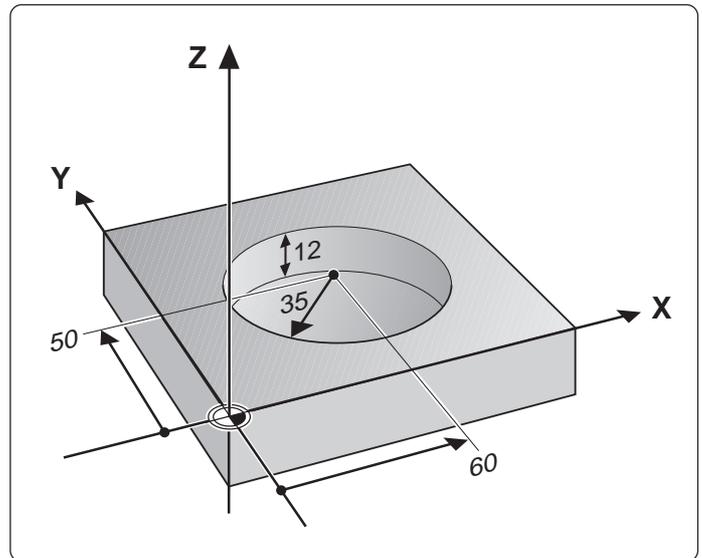
CIRCULAR POCKET cycle in a part program

```
%S814I G71 * ......................................... Start of program
N10 G30 G17 X+0 Y+0 Z–20 * .................................. Define workpiece blank
N20 G31 G90 X+100 Y+100 Z+0 *
N30 G99 T1 L+0 R+4 * ........................................... Define tool
N40 T1 G17 S2000 * ................................. Call tool
N50 G77 P01 –2 P02 –12 P03 –6 P04 80 P05 35
P06 100 * .................................................... Define circular pocket milling cycle
N60 G00 G40 G90 Z+100 M06 * .............................. Retract in the infeed axis, insert tool
N70 X+60 Y+50 M03 * ............................................ Approach the starting position (center of pocket), spindle ON
N80 Z+2 M99 * ....................................................... Pre-position in Z to setup clearance, cycle call
N90 Z+100 M02 * ................................................... Retract in the infeed axis, end of program
N99999 %S814I G71 *
```

8.3 SL Cycles (Group I)

SL cycles are highly efficient cycles that allow machining of any contour. These cycles have the following characteristics:

- A contour can be composed of several overlapping subcontours. Islands or pockets can form a subcontour.
- The subcontours are defined in subprograms.
- The control automatically superimposes the subcontours and calculates the points of intersection formed by overlapping.

The term **SL** is derived from the characteristic **S**ubcontour **L**ist of Cycle G37 CONTOUR GEOMETRY. Since this is purely a geometry cycle, no cutting data or feed values are defined.

The machining data are specified in the following cycles:

- PILOT DRILLING (G56)
- ROUGH-OUT (G57)
- CONTOUR MILLING (G58/G59)

The SL cycles of group II offer further, contour-oriented machining processes and are described later.

Each subprogram defines whether G41 or G42 radius compensation applies. The sequence of points determines the direction of rotation in which the contour is machined. The control infers from these data whether the specific subprogram describes a pocket or an island:

- The control recognizes a *pocket* if the tool path lies *inside* the contour
- The control recognizes an *island* if the tool path lies *outside* the contour

- The machining of the SL contour is determined by MP 7420.
- It is a good idea to run a graphic simulation before executing a program to see whether the contours were correctly defined.
- The memory capacity for programming an SL cycle is limited. For example, you can program 128 straight line blocks in one SL cycle.
- All coordinate transformations are allowed in programming the subcontours.
- Any words starting with F or M in the subprograms for the subcontours are ignored.

For easier familiarization, the following examples begin with only the rough-out cycle and then proceed progressively to the full range of functions provided by this group of cycles.

Programming parallel axes

Machining operations can also be programmed in parallel axes as SL cycles. (In this case, graphic simulation is not available.) The parallel axes must lie in the machining plane.

Input data

Parallel axes are programmed in the first coordinate block (positioning block, I,J,K block) of the first subprogram called in Cycle G37 CONTOUR GEOMETRY. Coordinate axes entered subsequently will be ignored.

CONTOUR GEOMETRY (G37)

Application

All subprograms that are superimposed to define the contour are listed in Cycle G37 CONTOUR GEOMETRY.

Input data

Enter the LABEL numbers of the subprograms. Up to 12 label numbers can be defined.

Activation

G37 becomes effective as soon as it is defined.
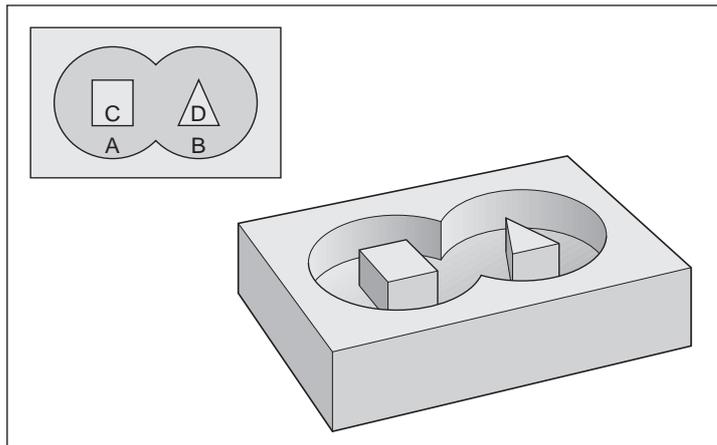
Fig. 8.14:   Example of an SL contour. *A* and *B* are pockets, *C* and *D* are islands

Example:

```
G99 T3 L+0 R+3.5 *
T3 G17 S1500 * ........................................................ Working plane perpendicular to Z axis
G37 P01 1 P02 2 P03 3 *
   .
   .
   .
G00 G40 Z+100 M2 *
   .
   .
   .
G98 L1 ....................................................................... First contour label for Cycle G37 CONTOUR GEOMETRY
G01 G42 X+0 Y+10 ................................................... Machining in the X/Y plane
X+20 Y+10
I+50 J+50
   .
   .
   .
```

ROUGH-OUT (G57)

The ROUGH-OUT cycle specifies cutting path and partitioning.

Sequence

- The control positions the tool in the tool axis over the first infeed point, taking the finishing allowance into account.
- The tool then penetrates the workpiece at the programmed feed rate for pecking.

Milling the contour:

- The tool mills the first subcontour at the specified feed rate, taking the finishing allowance into account.
- As soon as the tool returns to the infeed point, it is advanced to the next pecking depth.

This process is repeated until the programmed milling depth is reached.

- Further subcontours are milled in the same manner.

Roughing-out pockets:

- After milling the contour the pocket is roughed-out. The stepover is defined by the tool radius.
  Islands are jumped over.
- If required, pockets can be cleared with several downfeeds.
- At the end of the cycle, the tool is retracted to the setup clearance.

Required tool

The cycle requires a center-cut end mill (ISO 1641) if the pocket is not separately pilot drilled or if the tool must repeatedly jump over contours.

Input data

- SETUP CLEARANCE Ⓐ
- MILLING DEPTH Ⓑ
  The algebraic sign determines the working direction (a negative value means negative working direction).
- PECKING DEPTH Ⓒ
- FEED RATE FOR PECKING:
  Traversing speed of the tool during penetration
- FINISHING ALLOWANCE Ⓓ:
  Allowance in the machining plane (positive value)
- ROUGH-OUT ANGLE ⓐ:
  Feed direction for roughing-out.
  The rough-out angle is relative to the angle reference axis and can be set, so that the resulting cuts are as long as possible with few cutting movements.
- FEED RATE:
  Traversing speed of the tool in the machining plane

The machine parameters determine whether

- the contour is milled first and then surface machined, or vice versa
- the contour is milled conventionally or by climb cutting
- all pockets are roughed-out first and then contour-milled over all infeeds, or whether
- contour milling and roughing-out are performed mutually for each infeed
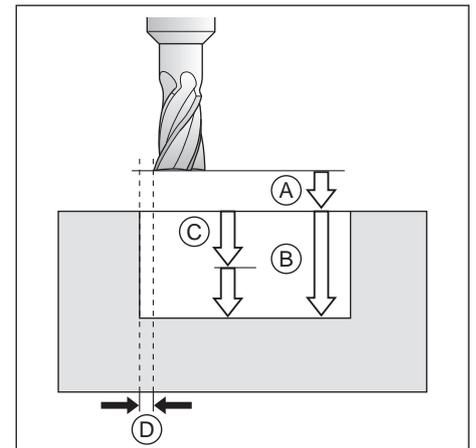
Fig. 8.15:   Infeeds and distances of the ROUGH-OUT cycle
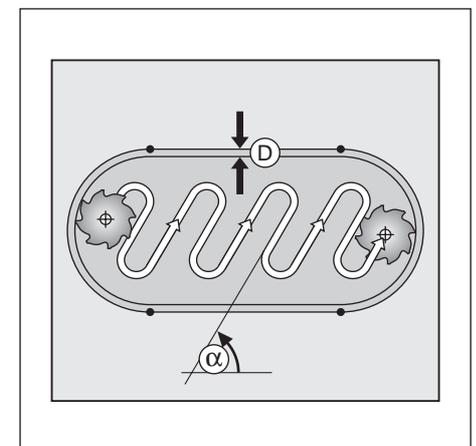
Fig. 8.16:   Cutter path for roughing-out

Example: Roughing-out a rectangular island

Rectangular island with rounded corners

Tool: center-cut end mill (ISO 1641),
radius 5 mm

Coordinates of the island corners:

	X		Y	
①	70	mm	60	mm
②	15	mm	60	mm
③	15	mm	20	mm
④	70	mm	20	mm

Coordinates of the auxiliary pocket:

	X		Y	
⑥	−5	mm	−5	mm
⑦	105	mm	−5	mm
⑧	105	mm	105	mm
⑨	−5	mm	105	mm

Starting point for machining:

⑤   X = 40 mm   Y = 60 mm

Setup clearance:	2	mm
Milling depth:	15	mm
Pecking depth:	8	mm
Feed rate for pecking:	100	mm/min
Finishing allowance:	0	
Rough-out angle:	0°	
Milling feed rate:	500	mm/min

G98 L1

G98 L2
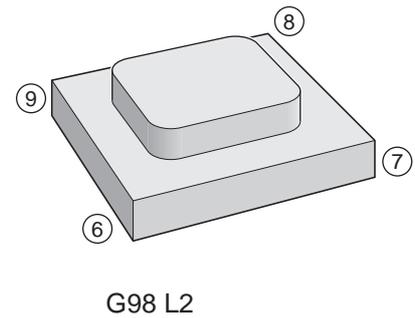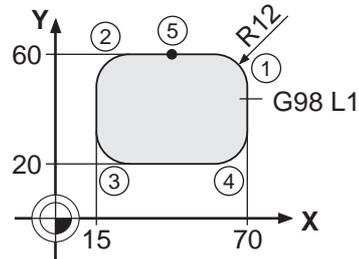
ROUGH-OUT cycle in a part program

```
%S818I G71 * ........................................ Start of program
N10 G30 G17 X+0 Y+0 Z–20 * ................................ Define workpiece blank
N20 G31 X+100 Y+100 Z+0 *
N30 G99 T1 L+0 R+3 * .............................................. Define tool
N40 T1 G17 S2500 * ............................................... Call tool
N50 G37 P01 2 P02 1 * ............................................ In the CONTOUR GEOMETRY cycle, state that the contour
                                                                    elements are described in subprograms 2 and 1
N60 G57 P01 –2 P02 –15 P03 –8 P04 100 P05 +0
P06 +0 P07 500 * ................................................. Cycle definition ROUGH-OUT
N70 G00 G40 G90 Z+100 M06 * ............................. Retract in the infeed axis, insert tool
N80 X+40 Y+50 M03 * ........................................... Pre-position in X/Y, spindle ON
N90 Z+2 M99 * ....................................................... Pre-position in Z to setup clearance, cycle call
N100 Z+100 M02 *

N110 G98 L1 *                                                      Subprogram 1:
N120 G01 G42 X+40 Y+60 *                                          Geometry of the island
N130 X+15 *                                                        (radius compensation G42 and machining in counterclockwise
                                                                    direction: the contour element is an island)

N150 Y+20 *
N160 G25 R12 *
N170 X+70 *
N180 G25 R12 *
N190 Y+60 *
N200 G25 R12 *
N210 X+40 *
N220 G98 L0 *

N230 G98 L2 *                                                      Subprogram 2:
N240 G01 G41 X–5 Y–5 *                                             Geometry of the auxiliary pocket:
N250 X+105 *                                                       External boundary of the area to
N260 Y+105 *                                                       be machined
N270 X–5 *                                                         (radius compensation G41 and machining in counterclockwise
N280 Y–5 *                                                         direction: the contour element is a pocket)
N290 G98 L0 *
N99999 %S818I G71 *
```

Overlapping contours

Pockets and islands can also be overlapped to form a new contour. The area of a pocket can thus be enlarged by another pocket or reduced by an island.

Starting position

Machining begins at the starting position of the first pocket listed in Cycle G37 CONTOUR GEOMETRY. The starting position should be located as far as possible from the superimposed contours.
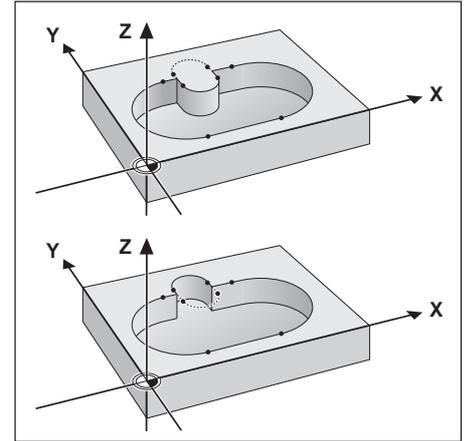
Fig. 8.17:   Examples of overlapping contours

Example: Overlapping pockets

The machining process starts with the first contour label defined in block 6. The first pocket must begin outside the second pocket.

Inside machining with a center-cut end mill (ISO 1641), tool radius 3 mm

Coordinates of the circle centers:

① X = 35 mm   Y = 50 mm
② X = 65 mm   Y = 50 mm

Circle radii

  R = 25 mm

Setup clearance:	2 mm
Milling depth:	10 mm
Pecking depth:	5 mm
Feed rate for pecking:	500 mm/min
Finishing allowance:	0
Rough-out angle:	0
Milling feed rate:	500 mm/min
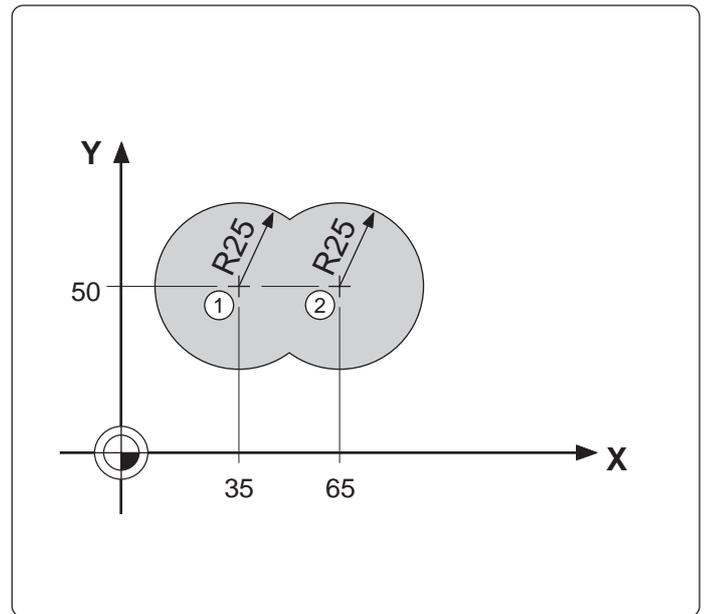
Cycle in a part program

```
%S820I G71 * ...................................................... Start of program
N10 G30 G17 X+0 Y+0 Z–20 * ................................. Define workpiece blank
N20 G31 X+100 Y+100 Z+0 *
N30 G99 T1 L+0 R+3 * ........................................... Define tool
N40 T1 G17 S2500 * ................................................ Call tool
N50 G37 P01 1 P02 2 * ........................................... In the CONTOUR GEOMETRY cycle, state that the contour
                                                                 elements are described in subprograms 1 and 2
N60 G57 P01 –2 P02 –15 P03 –8 P04 100 P05 +0
P06 +0 P07 500 * ..................................................... Cycle definition ROUGH-OUT
N70 G00 G40 G90 Z+100 M06 * ............................. Retract in the infeed axis, insert tool
N80 X+50 Y+50 M03 * ............................................. Pre-position in X/Y, spindle ON
N90 Z+2 M99 * ......................................................... Pre-position in Z to setup clearance, cycle call
N100 Z+100 M02 *

N110 G98 L1 *
       .
       .
       .
N140 G98 L0 *
N150 G98 L2 *
       .
       .
       .
N180 G98 L0 *
N99999 %S820I G71 *
```

Subprograms: Overlapping pockets

Pocket elements *A* and *B* overlap.

The control automatically calculates the points of intersection S_1 and S_2 (they do not have to be programmed). The pockets are programmed as full circles.

```
N110    G98 L1 *
N120    G01 G41 X+10 Y+50 *              A    Left pocket
N130    I+35 J+50 G03 X+10 Y+50 *
N140    G98 L0 *

N150    G98 L2 *
N160    G01 G41 X+90 Y+50 *              B    Right pocket
N170    I+65 J+50 G03 X+90 Y+50 *
N180    G98 L0 *
N99999 % S820I G71 *
```
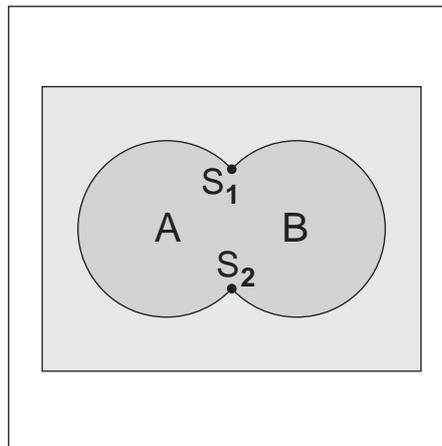

Fig. 8.18:   Points of intersection S_1 and S_2 of pockets *A* and *B*

Depending on the control setup (machine parameters), machining starts either with the outline or the surface:
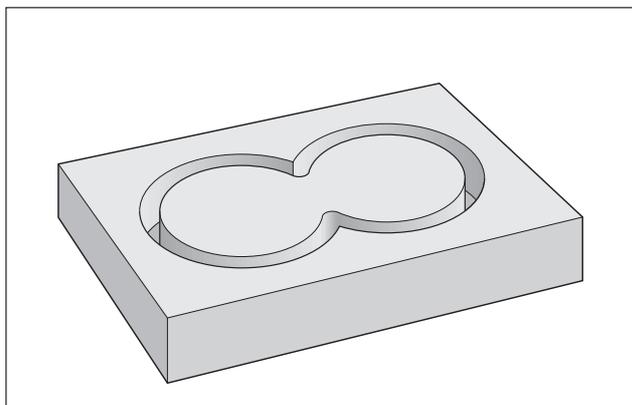
Fig. 8.19:   Outline is machined first
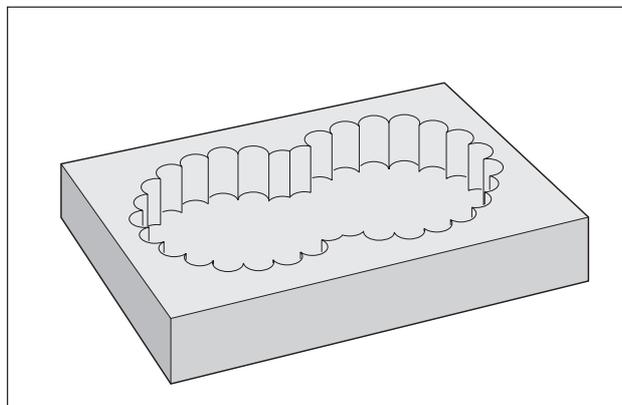
Fig. 8.20:   Surface is machined first

Area of inclusion

Both surfaces *A* and *B* are to be machined, including the mutually overlapped area.

- *A* and *B* must be pockets.
- The first pocket (in Cycle G37) must start outside the second pocket.

```
N110    G98 L1 *
N120    G01 G41 X+10 Y+50 *
N130    I+35 J+50 G03 X+10 Y+50 *
N140    G98 L0 *

N150    G98 L2 *
N160    G01 G41 X+90 Y+50 *
N170    I+65 J+50 G03 X+50 Y+50 *
N180    G98 L0 *
```
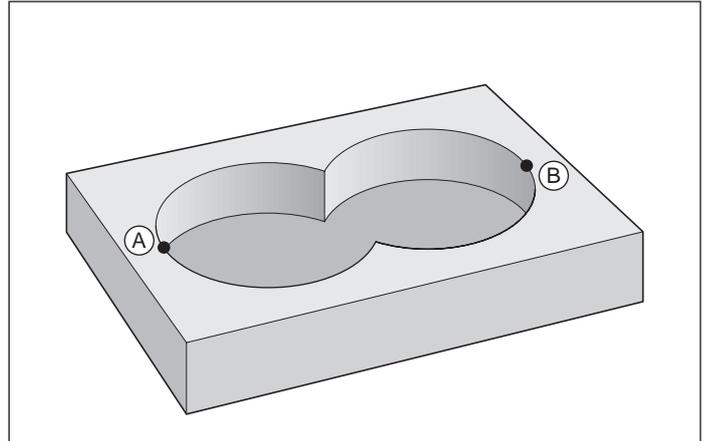

Fig. 8.21:   Overlapping pockets: area of inclusion

Area of exclusion

Surface *A* is to be machined without the portion overlapped by *B*.

- *A* must be a pocket and *B* an island.
- *A* must start outside of *B*.

```
N110    G98 L1 *
N120    G01 G41 X+10 Y+50 *
N130    I+35 J+50 G03 X+10 Y+50 *
N140    G98 L0 *

N150    G98 L2 *
N160    G01 G42 X+90 Y+50 *
N170    I+65 J+50 G03 X+90 Y+50 *
N180    G98 L0 *
```
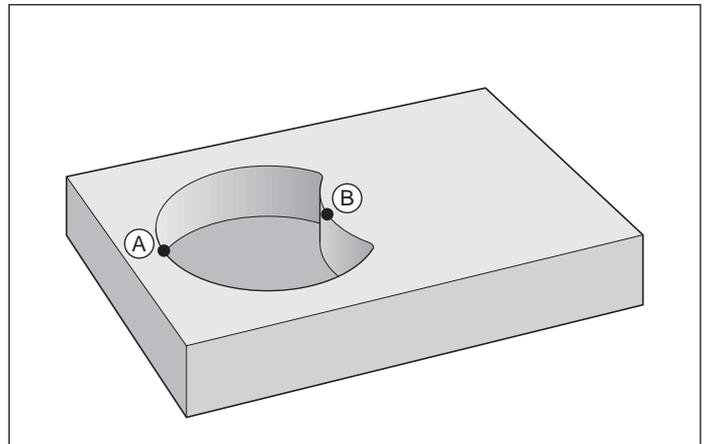

Fig. 8.22: Overlapping pockets: area of exclusion

Area of intersection

Only the area overlapped by both *A* and *B* is to be machined.

- *A* and *B* must be pockets.
- *A* must start inside *B*.

```
N110    G98 L1 *
N120    G01 G41 X+60 Y+50 *
N130    I+35 J+50 G03 X+60 Y+50 *
N140    G98 L0 *

N150    G98 L2 *
N160    G01 G41 X+90 Y+50 *
N170    I+65 J+50 G03 X+90 Y+50 *
N180    G98 L0 *
```
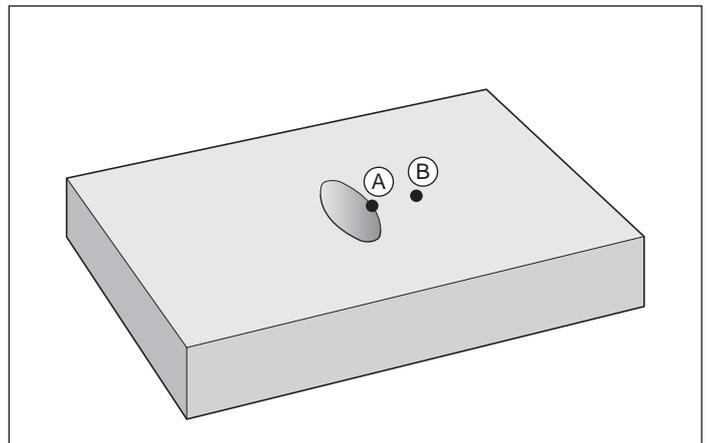

Fig. 8.23:   Overlapping pockets: area of intersection

The subprograms are used in the main program on page 8-22.

Subprogram: Overlapping islands

An island always requires a pocket as an additional boundary (here, G98 L1). A pocket can also reduce more than one island surface. The starting point of this pocket must be within the first island. The starting points of the remaining intersecting island contours must be outside the pocket.

```
%S822I G71 *
N10     G30 G17 X+0 Y+0 Z–20 *
N20     G31 X+100 Y+100 Z+0 *
N30     G99 T1 L+0 R+2.5 *
N40     T1 G17 S2500 *
N50     G37 P01 2 P02 3 P03 1 *
N60     G57 P01 –2 P02 –10 P03 –5 P04 100
        P05 +0 P06 +0 P07 500 *
N70     G00 G40 G90 Z+100 M06 *
N80     X+50 Y+50 M03 *
N90     Z+2 M99 *
N100    Z+100 M02 *
N110    G98 L1 *
N120    G01 G41 X+5 Y+5 *
N130    X+95 *
N140    Y+95 *
N150    X+5 *
N160    Y+5 *
N170    G98 L0 *
N180    G98 L2 *
          .
          .
          .
N210    G98 L0 *
N220    G98 L3 *
          .
          .
          .
N250    G98 L0 *
N99999 %S822I G71 *
```

Area of inclusion

Elements *A* and *B* are to be left unmachined, including the mutually overlapped surface.

• *A* and *B* must be islands.
• The first island must start outside the second island.

```
N180     G98 L2 *
N190     G01 G42 X+10 Y+50 *
N200     I+35 Y+50 G03 X+10 Y+50 *
N210     G98 L0 *
N220     G98 L3 *
N230     G01 G42 X+90 Y+50 *
N240     I+65 J+50 G03 X+90 Y+50 *
N250     G98 L0 *
N99999 % S822 I G71
```
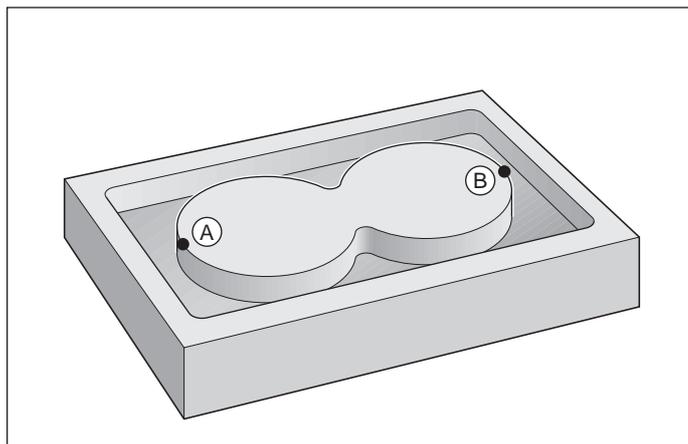

Fig. 8.24:   Overlapping islands: area of inclusion

✋ The subprograms and supplements are entered in the main program on page 8-24.

Area of exclusion

Surface *A* is to be left unmachined, without the portion overlapped by *B*.

- *A* must be an island and *B* a pocket.
- *B* must start within *A*.

```
N180    G98 L2 *
N190    G01 G42 X+10 Y+50 *
N200    I+35 J+50 G03 X+10 Y+50 *
N210    G98 L0 *
N220    G98 L3 *
N230    G01 G41 X+40 Y+50 *
N240    I+65 J+50 G03 X+40 Y+50 *
N250    G98 L0 *
N99999  S822I G71*
```
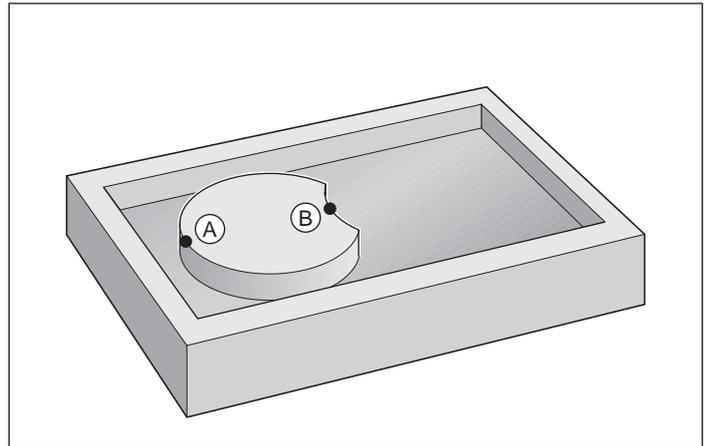

Fig. 8.25:   Overlapping islands: area of exclusion

Area of intersection

Only the area overlapped by both *A* and *B* is to remain unmachined.

- *A* and *B* must be islands.
- *A* must start within *B*.

```
N180    G98 L2 *
N190    G01 G42 X+60 Y+50 *
N200    I+35 J+50 G03 X+60 Y+50 *
N210    G98 L0 *
N220    G98 L3 *
N230    G01 G42 X+90 Y+50 *
N240    I+65 J+50 G03 X+90 Y+50 *
N250    G98 L0 *
N99999  % S822I G71
```
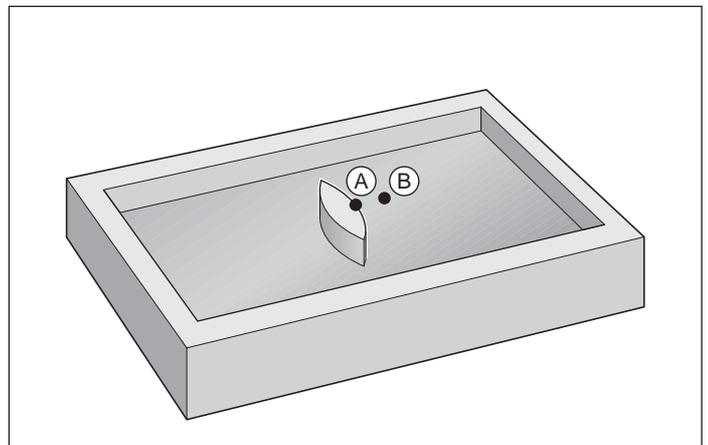

Fig. 8.26:   Overlapping islands: area of intersection

Example: Overlapping pockets and islands

PGM S824I is similar to PGM S820I but adds the islands *C* and *D*.

Tool: Center-cut end mill (ISO 1641), radius 3 mm

The contour is composed of the following elements:

Two overlapping pockets *(A* and *B)*, and two islands within the pockets *(C* and *D)*.

Cycle in a part program

```
%S824I G71 *
N10 G30 G17 X+0 Y+0 Z–20 *
N20 G31 X+100 Y+100 Z+0 *
N30 G99 T1 L+0 R+3 *
N40 T1 G17 S2500 *
N50 G37 P01 1 P02 2 P03 3 P04 4 *
N60 G57 P01 –2 P02 –10 P03 –5 P04 100 P05 +2 P06 +0 P07 500 *
N70 G00 G40 G90 Z+100 M06 *
N80 X+50 Y+50 M03 *
N90 Z+2 M99 *
N100 Z+100 M02 *

N110 G98 L1 *
N120 G01 G41 X+10 Y+50 *
N130 I+35 J+50 G03 X+10 Y+50 *
N140 G98 L0 *

N150 G98 L2 *
N160 G01 G41 X+90 Y+50 *
N170 I+65 J+50 G03 X+90 Y+50 *
N180 G98 L0 *

N190 G98 L3 *
N200 G01 G41 X+27 Y+42 *
N210 Y+58 *
N220 X+43 *
N230 Y+42 *
N240 X+27 *
N250 G98 L0 *

N260 G98 L4 *
N270 G01 G42 X+57 Y+42 *
N280 X+73 *
N290 X+65 Y+58 *
N300 X+57 Y+42 *
N310 G98 L0 *
N99999 %S824I G71 *
```

Fig. 8.27:   Milling of outline
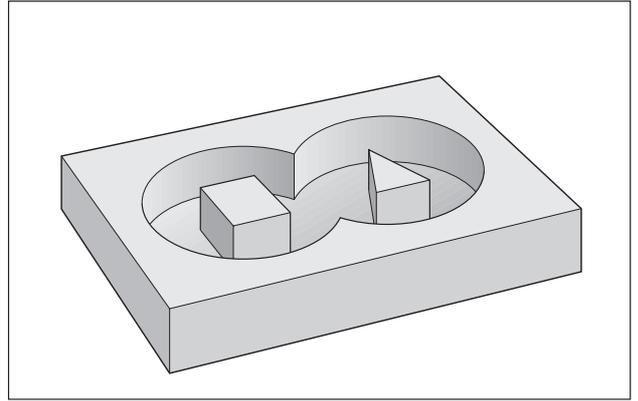
Fig. 8.28:   Finished workpiece

PILOT DRILLING (G56)

This cycle performs pilot drilling of holes for cutter infeed at the starting points of the subcontours. With SL contours consisting of several overlapping pockets and islands, the cutter infeed point is the starting point of the first subcontour:

- The tool is positioned at setup clearance over the first infeed point.
- The drilling sequence is identical to fixed Cycle G83 PECKING.
- The tool is then positioned above the second infeed point, and the drilling process is repeated.

Input data

- SETUP CLEARANCE
- TOTAL HOLE DEPTH
- PECKING DEPTH   } identical to Cycle G83 PECKING
- DWELL TIME
- FEED RATE

- FINISHING ALLOWANCE Ⓓ
  Allowed material for the drilling operation (see figure 8.30).
  The sum of the tool radius and the finishing allowance should be the same for pilot drilling as for roughing out.
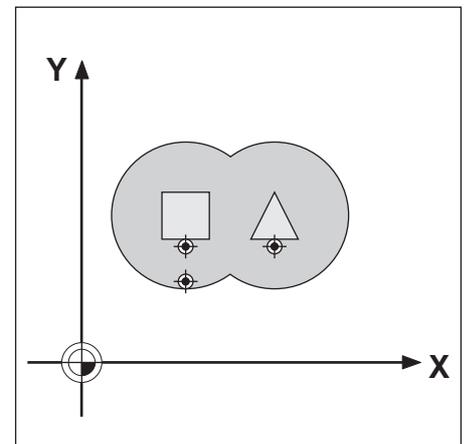
Fig. 8.29:   Example of cutter infeed points for PECKING
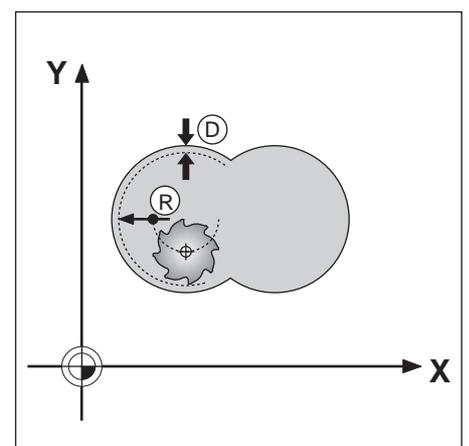
Fig. 8.30:   Finishing allowance

CONTOUR MILLING (G58/G59)

The CONTOUR MILLING cycles are used to finish-mill the contour pocket. The cycles can also be used generally for milling contours.

Sequence

- The tool is positioned at setup clearance over the first starting point.
- Moving at the programmed feed rate, the tool then penetrates to the first pecking depth.
- Upon reaching the first pecking depth, the tool mills the first contour at the programmed feed rate in the specified direction of rotation.
- At the infeed point, the control advances the tool to the next pecking depth.

This process is repeated until the programmed milling depth is reached. The remaining subcontours are milled in the same manner.

Required tool

The cycle requires a center-cut end mill (ISO 1641).

Direction of rotation during contour milling

Clockwise: G58
- For M3: up-cut milling for pocket and island

Counterclockwise: G59
- For M3: climb milling for pocket and island

Input data

- SETUP CLEARANCE Ⓐ
- MILLING DEPTH Ⓑ
  The algebraic sign determines the working direction (negative sign means negative working direction).
- PECKING DEPTH Ⓒ
- FEED RATE FOR PECKING:
  Traversing speed of the tool during penetration
- FEED RATE:
  Traversing speed of the tool in the machining plane
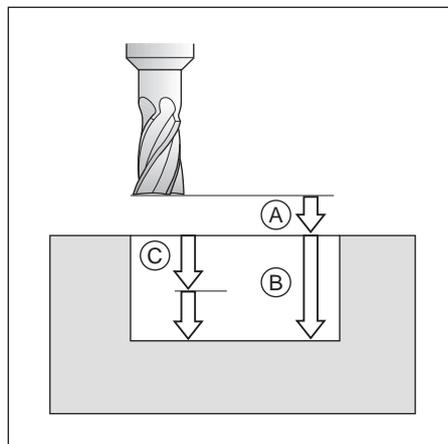
Fig. 8.31:   Infeeds and distances for CONTOUR MILLING
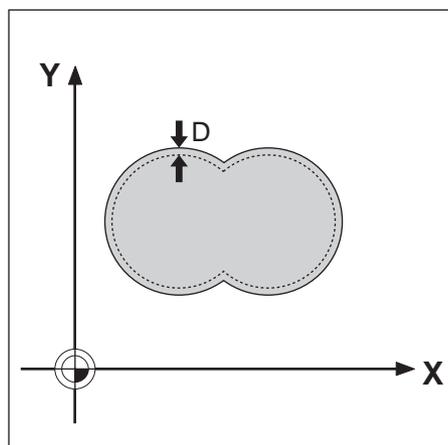
Fig. 8.32: Finishing allowance

8.3   SL Cycles (Group I)

The following scheme illustrates the application of the cycles PILOT DRILLING, ROUGH-OUT and CONTOUR MILLING in part programming.

1. List of contour subprograms

G37
No call

2. Drilling

Define and call the drilling tool
G56
Pre-positioning
Cycle call
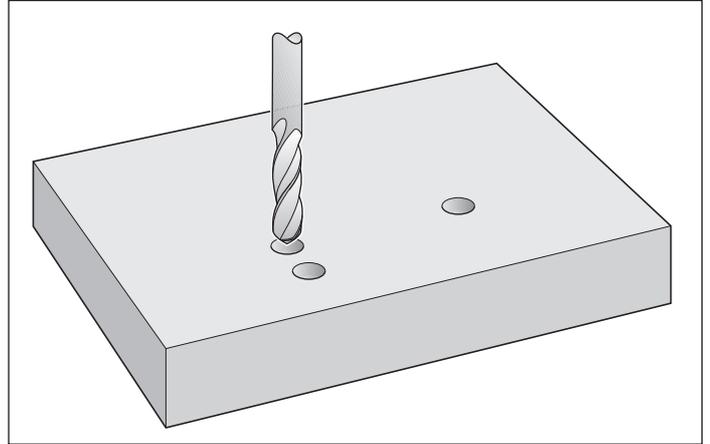
Fig. 8.33:   PILOT DRILLING cycle

3. Rough-out

Define and call rough milling tool
G57
Pre-positioning
Cycle call
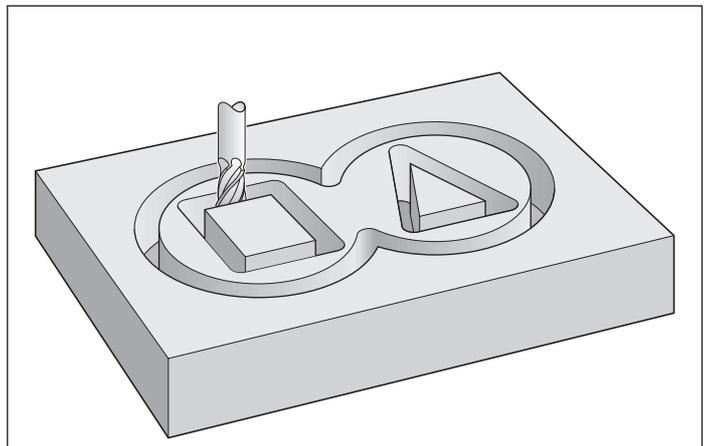
Fig. 8.34:   ROUGH-OUT cycle

4. Finishing

Define and call finish milling tool
G58/G59
Pre-positioning
Cycle call
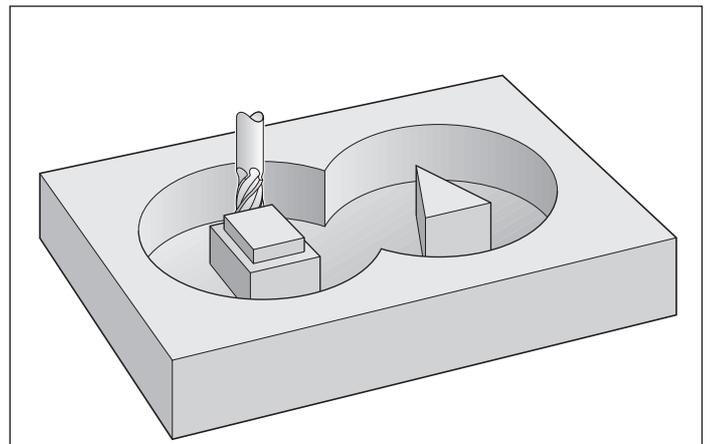
Fig. 8.35:   CONTOUR MILLING cycle

5. Contour subprograms

M02 *
Subprograms for the subcontours

Example: Overlapping pockets with islands

Inside machining with pilot drilling, roughing-out and finishing.

PGM S829I is based on S824I:

The main program section is expanded by the cycle definitions and calls for pilot drilling and finishing.

The contour subprograms 1 to 4 are identical to the ones in PGM S824I (see page 8-26) and are to be added after block N300.
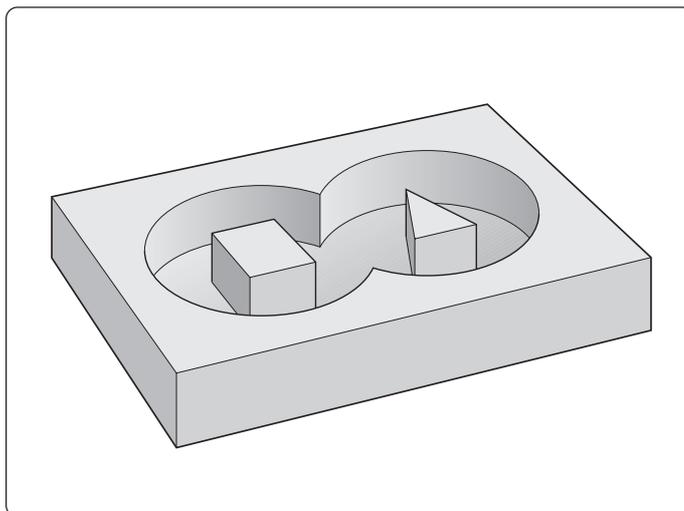
```
%S829I G71 * ........................................................ Start of program
N10 G30 G17 X+0 Y+0 Z–20 * ................................. Define workpiece blank
N20 G31 X+100 Y+100 Z+0 *
N30 G99 T1 L+0 R+2.5 * ......................................... Tool definition: drill bit
N40 G99 T2 L+0 R+3 * ........................................... Tool definition: roughing mill
N50 G99 T3 L+0 R+2.5 * ......................................... Tool definition: finishing mill
N60 L10,0 * ............................................................ Subprogram call for tool change
N70 G38 M06 * ...................................................... Program  STOP
N80 T1 G17 S2500 * .............................................. Tool call: drill bit
N90 G37 P01 1 P02 2 P03 3 P04 4 * ....................... Cycle definition: Contour Geometry
N100 G56 P01 –2 P02 –10 P03 –5 P04 500 P05 +2 * Cycle definition: Pilot Drilling
N110 Z+2 M03 *
N120 G79 * ............................................................ Cycle call: Pilot Drilling
N130 L10,0 *
N140 G38 M06 * .................................................... Tool  change
N150 T2 G17 S1750 * .............................................. Tool call: roughing mill
N160 G57 P01 –2 P02 –10 P03 –5 P04 100 P05+2
P06+0 P07 500 * ...................................................... Cycle definition: Rough-Out
N170 Z+2 M03 *
N180 G79 * ............................................................ Cycle call: Rough-Out
N190 L10,0 *
N200 G38 M06 * .................................................... Tool  change
N210 T3 G17 S2500 * .............................................. Tool call: finishing mill
N220 G58 P01 –2 P02 –10 P03 –10 P04 100
P05 500 * ................................................................ Cycle definition: Contour Milling
N230 Z+2 M03 *
N240 G79 * ............................................................ Cycle call: Contour Milling
N250 Z+100 M02 *

N260 G98 L10 * ...................................................... Subprogram for tool change
N270 T0 G17 *
N280 G00 G40 G90 Z+100 *
N290 X–20 Y–20 *
N300 G98 L0 *

From block N310: Add subprograms on page 8-26

N99999 %S829I G71 *
```

8.4 SL Cycles (Group II)

The SL cycles of group II allow *contour-oriented* machining of complex contours and achieve a particularly high degree of surface finish.

These cycles differ from those of group I in the following ways:

- Before the cycle starts, the TNC automatically positions the tool to the setup clearance.
- Each level of infeed depth is milled without interruptions since the cutter traverses around islands instead of over them.
- The radius of "inside corners" can be programmed — the tool keeps moving to prevent surface blemishes at inside corners (this applies for the outermost pass in Cycles G123 and G124).
- The contour is approached in a tangential arc for side finishing.
- For floor finishing, the tool again approaches the workpiece in a tangential arc (for tool axis Z, for example, the arc may be in the Z/X plane).
- The contour is machined throughout in either climb or up-cut milling.
- MP 7420 is replaced by DIRECTION OF ROTATION Q9.

The machining data (such as milling depth, finishing allowance and setup clearance) are entered as CONTOUR DATA in Cycle G120.

There are four cycles for contour-oriented machining:

- PILOT DRILLING (G121)
- ROUGH-OUT (G122)
- FLOOR FINISHING (G123)
- SIDE FINISHING (G124)

CONTOUR DATA (G120)

Application

Machining data for the subprograms describing the subcontours are entered in Cycle G120. These data are valid for Cycles G121 to G124.

Input data

- MILLING DEPTH Q1
  Distance between workpiece surface and pocket floor. The algebraic sign determines the working direction (negative sign means negative working direction).
- PATH OVERLAP FACTOR Q2
  Q2 * tool radius = stepover factor *k*
- ALLOWANCE FOR SIDE Q3
  Finishing allowance in the working plane
- ALLOWANCE FOR FLOOR Q4
  Finishing allowance in the tool axis
- WORKPIECE SURFACE COORDINATE Q5
  Absolute coordinate of the workpiece surface referenced to the work-piece datum
- SETUP CLEARANCE Q6
  Distance between the tool tip and the workpiece surface
- CLEARANCE HEIGHT Q7
  Absolute height at which the tool cannot collide with the workpiece (for intermediate positioning and retraction at the end of the cycle).
- INSIDE CORNER RADIUS Q8
  Inside "corner" rounding radius
- DIRECTION OF ROTATION Q9
  Direction of rotation for pockets:
     Clockwise (Q9 = –1)
     up-cut milling for pocket and island
     Counterclockwise (Q9 = +1)
     climb milling for pocket and island

Activation

G120 becomes effective immediately upon definition.

The machining parameters can be checked during a program interruption and overwritten if required.

If the SL cycles are used in Q parameter programs, the cycle parameters Q1 to Q17 cannot be used as program parameters.
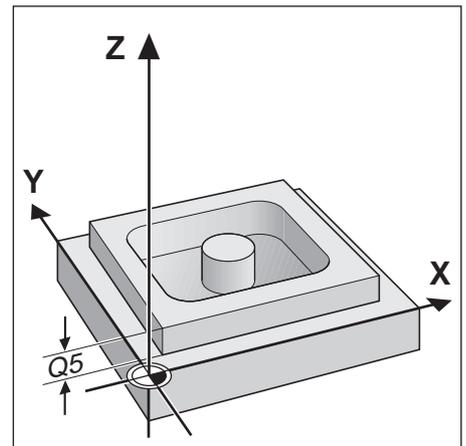
Fig. 8.36: Workpiece surface coordinate Q5
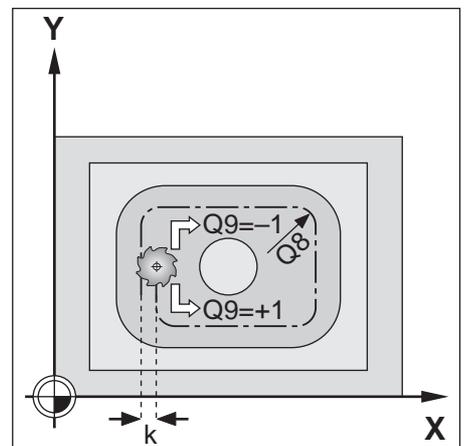
Fig. 8.37:   Direction of rotation Q9 and stepover factor k
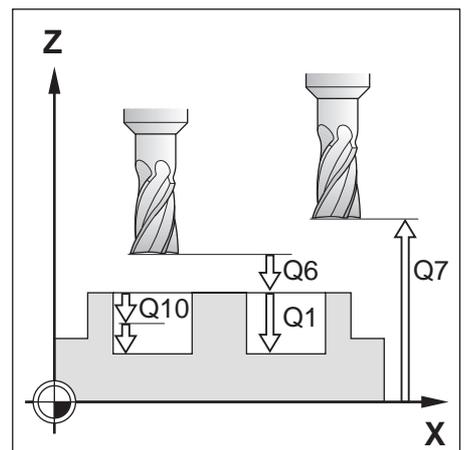
Fig. 8.38:   Distance and infeed parameters

8.4   SL Cycles (Group II)

PILOT DRILLING (G121)

Application

Cycle G121 is for PILOT DRILLING of the cutter infeed points. It accounts for the ALLOWANCE FOR SIDE and the ALLOWANCE FOR FLOOR as well as the radius of the rough-out tool. The cutter infeed points also serve as starting points for milling.

Sequence

Same as Cycle G83 PECKING.

Input data

- PECKING DEPTH Q10
  Dimension by which the tool drills in each infeed (negative sign for negative direction)
- FEED RATE FOR PECKING Q11
  Traversing speed of the tool in mm/min during drilling
- ROUGH-OUT TOOL NUMBER Q13
  Tool number of the roughing mill
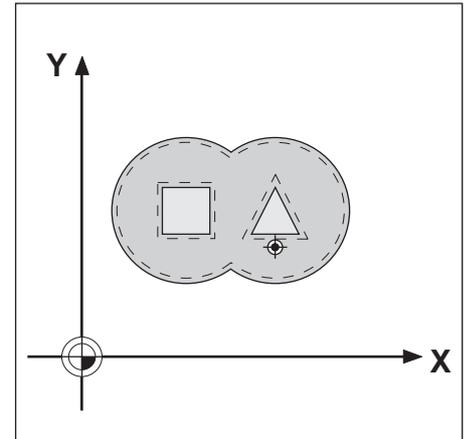
Fig. 8.39:   Possible infeed point for PILOT DRILLING

ROUGH-OUT (G122)

Sequence

- The control positions the tool over the cutter infeed point
- The ALLOWANCE FOR SIDE is taken into account.
- After reaching the first pecking depth, the tool mills the contour in an outward direction at the programmed feed rate Q12.
- First the island contours (*C* and *D* in figure 8.40) are rough-milled until the pocket contour *(A, B)* is approached.
- Then the pocket contour is rough-milled and the tool is retracted to the CLEARANCE HEIGHT.

Input data

- PECKING DEPTH Q10
  Dimension by which the tool is plunged in each infeed (negative sign for negative direction)
- FEED RATE FOR PECKING Q11
  Traversing speed of the tool in mm/min during penetration
- FEED RATE FOR MILLING Q12
  Traversing speed of the tool in mm/min while milling

Required tool

The cycle requires a center-cut end mill (ISO 1641).
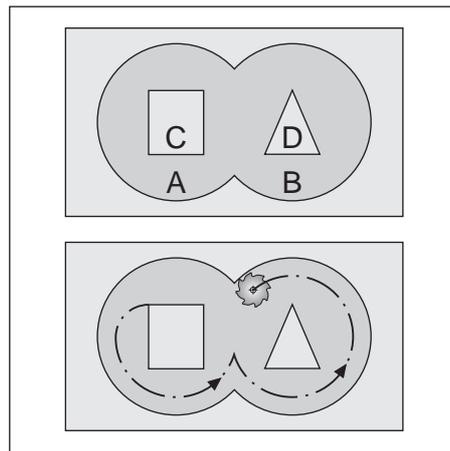
Fig. 8.40:   Cutter path for ROUGH-OUT. *A* and *B* are pockets, *C* and *D* are islands

FLOOR FINISHING (G123)

Sequence

Cycle G123 FLOOR FINISHING functions similar to Cycle G122 ROUGH-OUT. The tool approaches the machining plane in a vertically tangential arc.

Input data

- FEED RATE FOR PECKING Q11
  Traversing speed of the tool during penetration
- FEED RATE FOR MILLING Q12
 Traversing speed of the tool in the machining plane

SIDE FINISHING (G124)

Sequence

The subcontours are approached and departed on a tangential arc. Each subcontour is finish-milled separately.

Input data

- DIRECTION OF ROTATION Q9
  Direction of the cutter path
  Clockwise: +1
  Counterclockwise: –1
- PECKING DEPTH Q10
  Dimension by which the tool plunges in each infeed
- FEED RATE FOR PECKING Q11
  Traversing speed during penetration
- FEED RATE FOR MILLING Q12
  Traversing speed for milling
- ALLOWANCE FOR SIDE Q14
  Enter the allowed material for several finish-milling operations.
  If Q14 = 0 is entered, the remaining finishing allowance will be cleared.

Prerequisites

- The sum of ALLOWANCE FOR SIDE (Q14) and the radius of the finish mill must be smaller than sum of ALLOWANCE FOR SIDE (Q3, Cycle G120) and the radius of the roughing mill. This calculation also holds if G124 is run without having roughed out with G122, in which case 0 should be used for the radius of the roughing mill.

Example: Rectangular pocket with round island

Input parameters:

Milling depth Q1:	–15 mm
Path overlap Q2:	1
Allowance side Q3:	1 mm
Allowance depth Q4:	1 mm
Top surface of workpiece Q5:	0
Setup clearance Q6:	2 mm
Clearance height Q7:	50
Rounding radius Q8:	10 mm
Direction of rotation Q9:	+1

Subcontours are defined in subprograms 1 and 2.
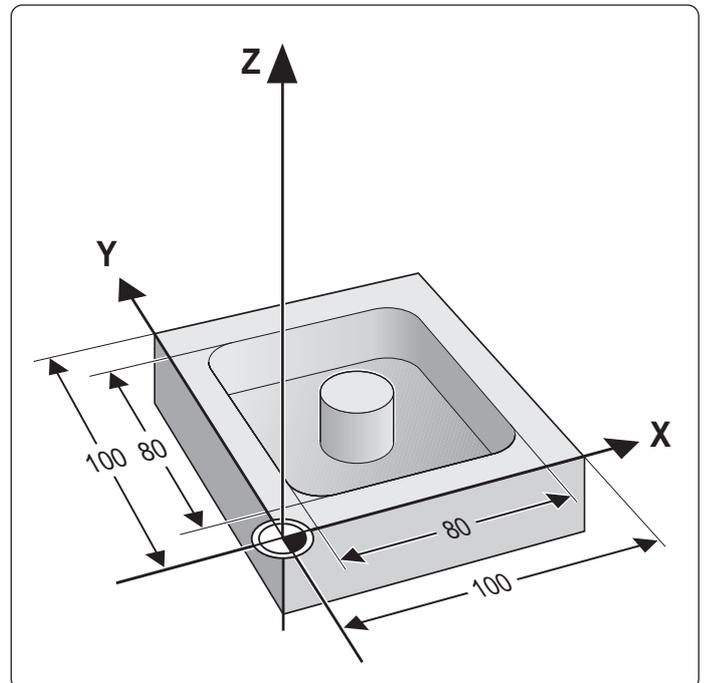
Part program

%S835I G71 *	Start of program
N10 G30 G17 X+0 Y+0 Z–20 *	Define workpiece blank
N20 G31 G90 X+100 Y+100 Z+0 *	
N30 G99 T1 L+0 R+3 *	Define tools
N40 G99 T2 L+0 R+2.5 *	
N50 G99 T3 L+0 R+2.5 *	
N60 G37 P01 1 P02 2 *	Cycle definition: Contour Geometry
N70 G120 Q1=–15 Q2=1 Q3=+1 Q4=+1 Q5=+0	
Q6=–2 Q7=+50 Q8=+10 Q9=+1 *	Cycle definition: Contour Data
N80 L10,0 *	Call subprogram for tool change
N90 T1 G17 S2500 *	
N100 G121 Q10=–10 Q11=100 Q13=2 *	Cycle definition: Pilot Drilling
N110 G79 M3 *	Cycle call: Pilot Drilling
N120 L10,0 *	Call subprogram for tool change
N130 T2 G17 S1500 *	
N140 G122 Q10=–10 Q11=100 Q12=500 *	Cycle definition: Rough-Out
N150 G79 M3 *	Cycle call: Rough-Out
N160 L10,0 *	Call subprogram for tool change
N170 T3 G17 S3000 *	
N180 G123 Q11=80 Q12=250 *	Cycle definition: Floor Finishing
N190 G79 M3 *	Cycle call: Floor Finishing
N200 G124 Q9=+1 Q10=–5 Q11=100 Q12=240	
Q14=+0 *	Cycle definition: Side Finishing
N210 G79 M3 *	Cycle call: Side Finishing
N220 G00 G40 Z+100 M2 *	
N230 G98 L10 *	Subprogram for tool change
N240 T0 G17 *	
N250 G00 G40 G90 Z+100 *	
N260 X–20 Y–20 M6 *	
N270 G98 L0 *	
N280 G98 L1 *	Contour subprogram: Rectangular Pocket
N290 G01 G42 X+10 Y+50 *	
N300 Y+90 *	
N310 X+90 *	
N320 Y+10 *	
N330 X+10 *	
N340 Y+50 *	
N350 G98 L0 *	
N360 G98 L2 *	Contour subprogram: Circular Island
N370 G01 G41 X+35 Y+50 *	
N380 I+50 J+50 *	
N390 G02 X+35 Y+50 *	
N400 G98 L0 *	
N99999 %S835I G71 *	

CONTOUR TRAIN (G125)

Sequence

This cycle facilitates the machining of open contours (the starting point of the contour is not the same as its end point).

G125 CONTOUR TRAIN offers considerable advantages over machining an open contour using positioning blocks:

- The control monitors the operation to prevent undercuts and surface blemishes. It is recommended that you run a graphic simulation of the contour before execution.

- If the radius of the selected tool is too large, the corners of the contour may have to be reworked.

- The contour can be machined throughout by up-cut or by climb milling.

- The tool can be traversed back and forth for milling in several infeeds. This results in faster machining.

- Allowance values can be entered in order to perform repeated rough-milling and finish milling operations.
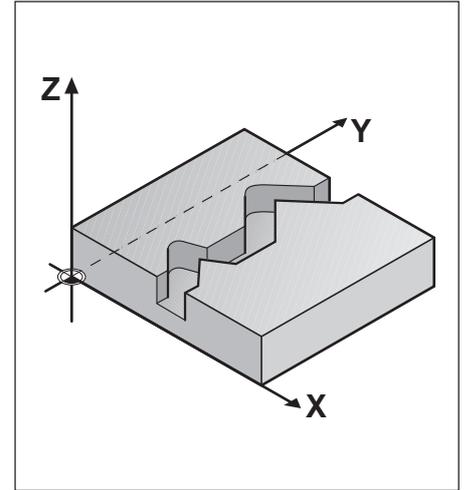
Fig. 8.41:   Example of an open contour

> G125 CONTOUR TRAIN should not be used for closed contours. With closed contours, the starting point and end point of the contour must not be located in a contour corner.

Input data

- MILLING DEPTH Q1
  Distance between workpiece surface and contour floor. The sign determines the working direction (a negative sign means negative working direction).

- ALLOWANCE FOR SIDE Q3
  Finishing allowance in the machining plane

- WORKPIECE SURFACE COORDINATE Q5
  Absolute coordinate of the workpiece surface referenced to the work-piece datum

- CLEARANCE HEIGHT Q7
  Absolute height at which the tool cannot collide with the workpiece. Position for tool retraction at the end of the cycle.

- PECKING DEPTH Q10
  Dimension by which the tool is plunged for each infeed

- FEED RATE FOR PECKING Q11
  Traversing speed of the tool in the tool axis

- FEED RATE FOR MILLING Q12
  Traversing speed of the tool in the machining plane

- CLIMB OR UP-CUT Q15
  Climb milling: input value =   +1
  Up-cut milling: input value =   – 1
  To enable climb milling and conventional up-cut milling alternately in several infeeds: input value = 0

> - If Cycle G125 CONTOUR TRAIN is used, only the first label from Cycle G37 CONTOUR GEOMETRY will be processed.
> - Each subprogram can contain up to 128 contour elements.
> - Cycle G120 CONTOUR DATA is not required.

Example

Input parameters in Cycle G125:

Milling depth Q1:	–12 mm
Allowance for side Q3:	0
Top surface of workpiece Q5:	0
Clearance height Q7:	10
Pecking depth Q10:	–2 mm
Feed rate for pecking Q11:	100 mm/min
Feed rate for milling Q12:	200 mm/min
Milling type Q15 (climb milling):	+1
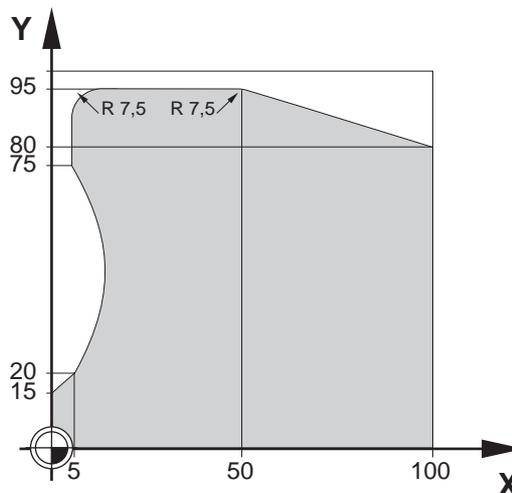
Cycle in part program

```
%S837I G71 * ........................................... Start of program
N10 G30 G17 X+0 Y+0 Z–20 * .................................... Define workpiece blank
N20 G31 G90 X+100 Y+100 Z+0 *
N30 G99 T1 L+0 R+10 * ............................................ Define tool
N40 T1 G17 S1500 * ................................................ Call tool
N50 G37 P01 1 * ...................................................... Cycle definition: Contour Geometry
N60 G125 Q1=–12 Q3=+0 Q5=+0 Q7=+10 Q10=–2
Q11=100 Q12=200 Q15=+1 * .................................. Cycle definition: Contour Train
N70 G00 G40 G90 Z+100 M3 * ................................ Retract in the infeed axis, spindle ON
N80 G79 * ................................................................. Cycle call
N90 G00 G40 Z+100 M2 *

N100 G98 L1 * .......................................................... Contour subprogram
N110 G01 G41 X+0 Y+15 *
N120 X+5 Y+20 *
N130 G06 X+5 Y+75 *
N140 G01 Y+95 *
N150 G25 R7.5 *
N160 G01 X+50 *
N170 G25 R7.5 *
N180 X+100 Y+80 *
N190 G98 L0 *
N99999 %S837I G71 *
```

8.4   SL Cycles (Group II)

CYLINDER SURFACE G127

The machine tool and the TNC must have been prepared by the machine manufacturer for the use of Cycle G127.

Process

This cycle enables you to program a contour in two dimensions and then roll it onto a cylindrical surface for 3D machining.

The contour is described in a subprogram identified in Cycle G37 (CONTOUR GEOM.).

The subprogram contains coordinates in a rotary axis and in its parallel axis. The rotary axis C, for example, has Z as its parallel axis. The path functions G01, G02/G03 with R, G25 are available for programming.

The dimensions in the rotary axis can be entered as desired either in degrees or in mm (inch). You can select the desired dimension type in the cycle definition.
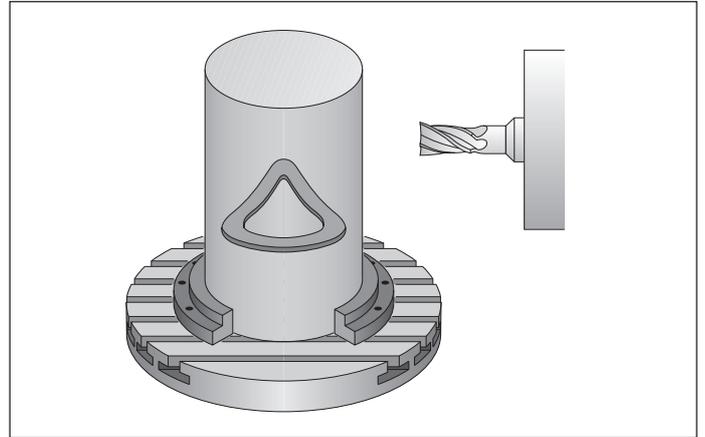
Fig. 8.42:   Contour on a cylindrical surface
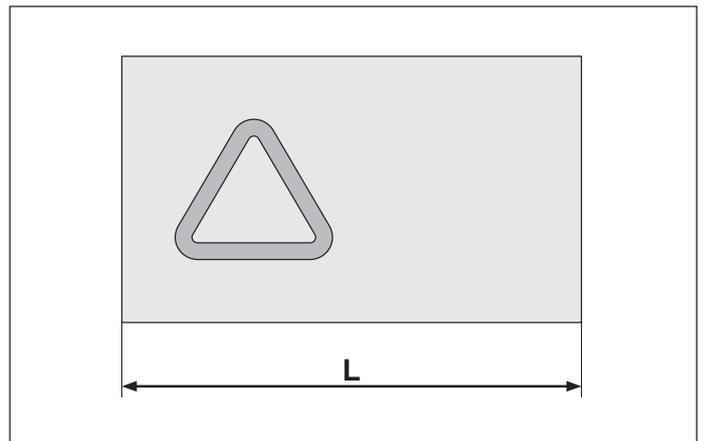
Fig. 8.43:   The unrolled surface of the cylinder:
L = Diameter of cylinder * 3.14

G127 CYLINDER SURFACE should not be used for closed contours. With closed contours, the starting point and end point of the contour must not be located in a contour corner.

Input

- MILLING DEPTH Q1:
  Distance between the cylindrical surface and the floor of the contour.

- ALLOWANCE FOR SIDE Q3:
  Finishing allowance in the plane of the unrolled cylindrical surface.
  This allowance is effective in the direction of the radius compensation.

- SETUP CLEARANCE Q6:
  Distance between the tool tip and the cylinder surface.

- PECKING DEPTH Q10:
  Dimension by which the tool advances in each infeed.

- FEED RATE FOR PECKING Q11:
  Feed rate for traversing in the tool axis.

- FEED RATE FOR MILLING Q12:
  Feed rate for traversing in the working plane.

- RADIUS Q16:
  Radius of the cylinder on which the contour is to be machined.

- DIMENSION TYPE (ANG/LIN) Q17:
  The dimensions for the rotary axis of the subprogram are given either
  in degrees (0) or in mm/inch (1).

Prerequisites

- The cycle requires a center-cut end mill (ISO 1641).

- The cylinder must be set up centered on the rotary table.

- The tool axis must be perpendicular to the rotary table. If this is not the
  case, an error message will result.

Example:

Rectangular channel on a cylindrical surface

Channel width	20 mm
Channel height	60 mm
Depth	7.5 mm
Cylinder diameter	50 mm

Length of programming plane =
Circumference of the cylinder:
3.14 x 50 mm = 157 mm

You wish to enter the dimensional data for
the rotary axis in mm (Q17 = 1)!
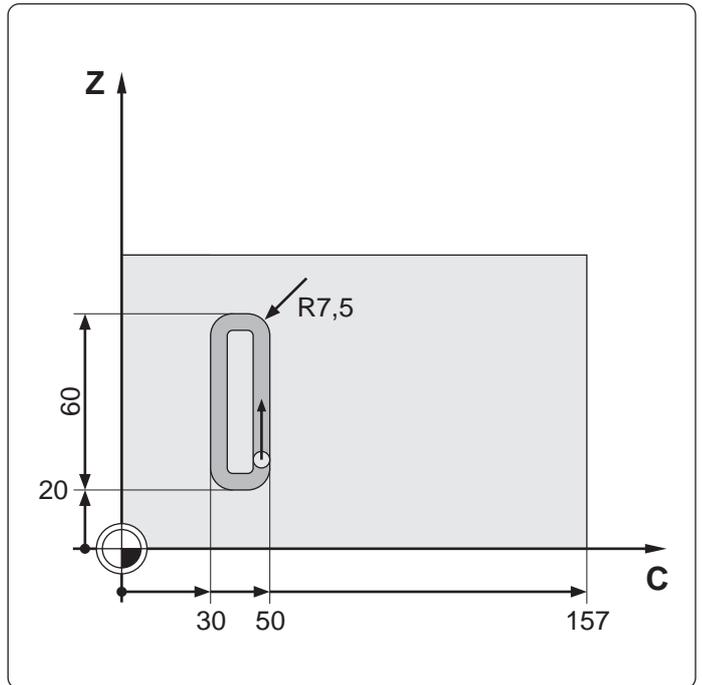
Cycle in a part program

```
%CYLSURF G71*
N10   G30 G18 X+0 Y+0 Z–20*
N20   G31 G90 X+100 Y+100 Z+0*
N30   G99 T1 L+0 R+6*
N40   T1 G18 S100*
N50   G00 G40 G90 Y+100*
N60   G127........................................................Cycle definition CYLINDER SURFACE
      Q1=–7.5 .......................................................;MILLING DEPTH
      Q3=+0 ..........................................................;ALLOWANCE FOR SIDE
      Q6=+2 ..........................................................;SETUP CLEARANCE
      Q10=+4 .........................................................;PECKING DEPTH
      Q11=100 .......................................................;FEED RATE FOR PECKING
      Q12=250 .......................................................;FEED RATE FOR MILLING
      Q16=+25 .......................................................;RADIUS
      Q17=1* ..........................................................;DIMENSION TYPE (ANG/LIN)
N70   C+0 M3 * .....................................................Pre-position on the rotary axis
N80   G79* ...........................................................Call the cycle
N90   G00 Y+200 M2 * ...........................................Retract, end of main program

N100 G98 L1*
N110 G01 G41 C+40 Z+20 * ...................................Start position C 40 mm
N120 C+50 Z+20*
N130 G25 R7.5*
N140 G91 Z+60*
N150 G90 G25 R7.5*
N160 G91 C–20*
N170 G90 G25 R7.5*
N180 L Z+20
N190 G25 R7.5*
N200 C+40*
N210 G98 L0 * ......................................................End of subprogram
N99999 % CYLSURF G71*
```

8.5   Coordinate Transformations

Once a contour has been programmed, it can be positioned on the workpiece at various locations and in different sizes through the use of coordinate transformations. The following cycles are available for this:

- DATUM SHIFT (G53/G54)
- MIRROR IMAGE (G28)
- ROTATION (G73)
- SCALING (G72)

The original contour must be marked in the part program as a subprogram or a program section.

Duration of effect

A coordinate transformation becomes effective as soon as it is defined, and remains in effect until it is changed or cancelled.

Cancellation

Coordinate transformations can be cancelled in the following ways:

- Define cycles for basic behavior with a new value (such as scaling factor 1)
- Execute a miscellaneous function M02 or M30, or an N99999 %... block (depending on machine parameters)
- Select a new program
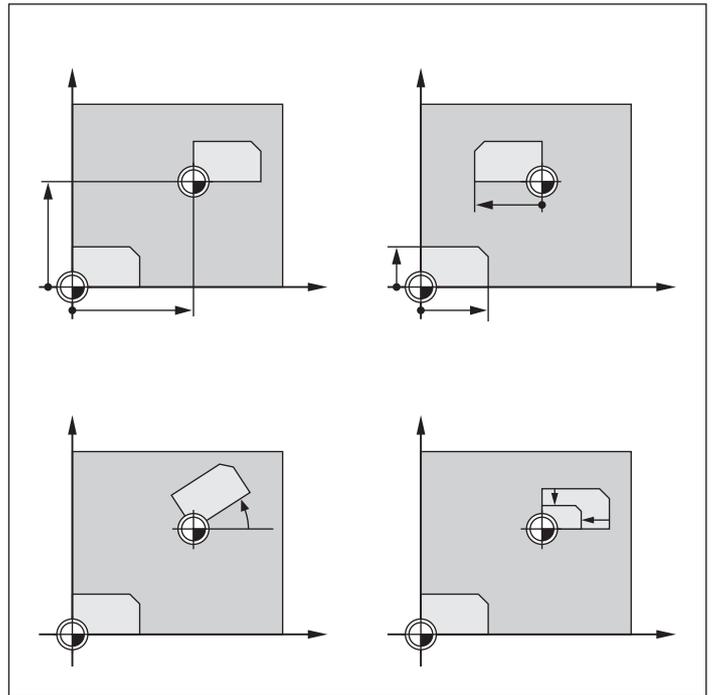
Fig. 8.44:   Examples of coordinate transformations

DATUM SHIFT (G54)

Application

A datum shift allows machining operations to be repeated at various locations on the workpiece.

Activation

When the DATUM SHIFT cycle is defined, all coordinate data are based on the new datum. The datum shift is shown in the additional status display.

Input data

For a datum shift, you need only enter the coordinates of the new datum (zero point). Absolute values are referenced to the manually set workpiece datum. Incremental values are referenced to the datum which was last valid (this can be a datum which has already been shifted).
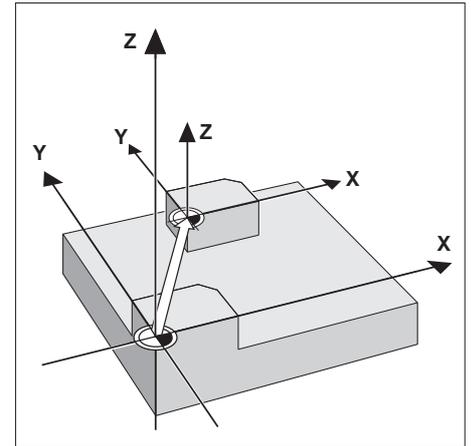
Fig. 8.45:   Activation of datum shift
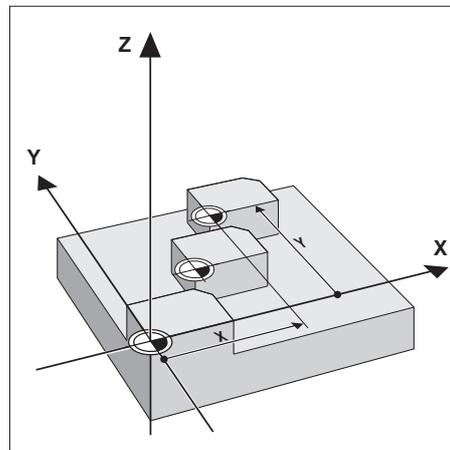
Fig. 8.46: Datum shift, absolute
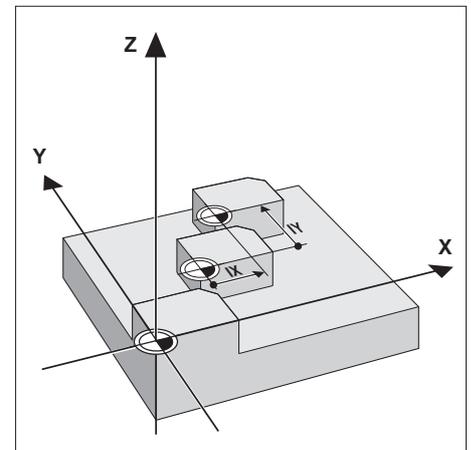
Fig. 8.47: Datum shift, incremental

Cancellation

A datum shift is cancelled by entering the datum shift coordinates X = 0, Y = 0 and Z = 0.

When combining transformations, a datum shift must be programmed before the other transformations.

Graphics

If you program a new workpiece blank after a datum shift, MP 7310 determines whether the workpiece blank is referenced to the current datum or the original datum (MP 7310: see page 11-11). Referencing a new workpiece blank to the current datum enables you to display each part in a program in which several parts are machined.

Example: Datum shift

A machining sequence in the form of a sub-
program is to be executed twice:

a)   once, referenced to the specified datum
① X+0/Y+0, and
b)   a second time, referenced to the shifted
datum ② X+40/Y+60.
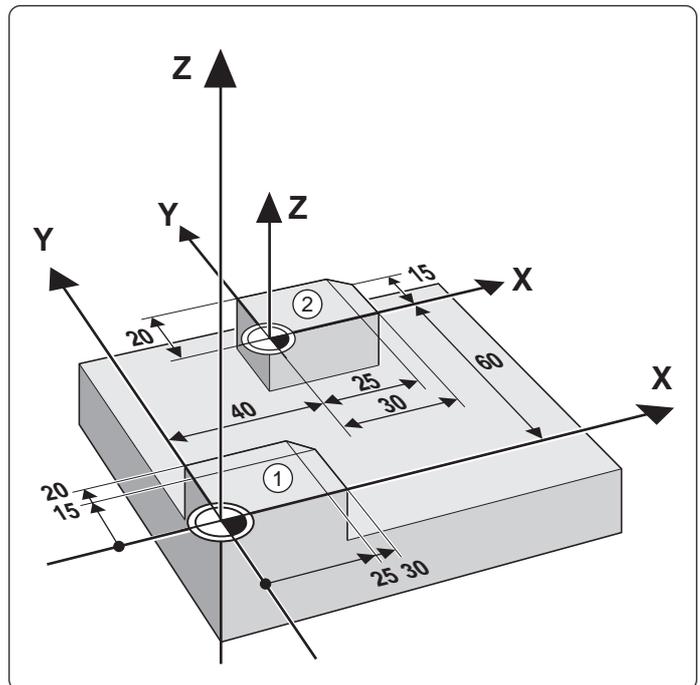
Cycle in part program

```
%S840I G71 * ........................................................... Start of program
N10 G30 G17 X+0 Y+0 Z–20 * .................................. Define workpiece blank
N20 G31 X+100 Y+100 Z+0 *
N30 G99 T1 L+0 R+4 * ............................................ Define tool
N40 T1 G17 S1500 * ................................................. Call tool
N50 G00 G40 G90 Z+100 * ...................................... Retract in the infeed axis
N60 L1,0 * .................................................................. Version 1 without datum shift
N70 G54 X+40 Y+60 *
N80 L1,0 * .................................................................. Version 2 with datum shift
N90 G54 X+0 Y+0 * ................................................... Cancel datum shift
N100 Z+100 M02 *
N110 G98 L1 *
            .
            .
            .
N230 G98 L0 *
N99999 %S840I G71 *
```

> **Subprogram**
>
> N110 G98 L1 *
> N120 X–10 Y–10 M03 *
> N130 Z+2 *
> N140 G01 Z–5 F200 *
> N150 G41 X+0 Y+0 *
> N160 Y+20 *
> N170 X+25 *
> N180 X+30 Y+15 *
> N190 Y+0 *
> N200 X+0 *
> N210 G40 X–10 Y–10 *
> N220 G00 Z+2 *
> N230 G98 L0 *

Depending on the transformations, the subprogram is added to the program at the following positions (NC blocks):

	LBL 1	**LBL 0**
Datum shift	block N110	block N230
Mirror image, rotation, scaling	block N130	block N250

DATUM SHIFT with datum tables (G53)

Application

Datum tables are applied for

- frequently repeating machining sequences at various locations on the workpiece
- frequent use of the same datum shift

The datum points from datum tables are only effective with absolute coordinate values.

Within a program, datum points can either be programmed directly in the cycle definition or called from a datum table.

Fig. 8.48:   Similar datum shifts

Input

Enter the number of the datum from the datum table or a Q parameter number. If you enter a Q parameter number, the TNC activates the datum number found in the Q parameter.

Cancellation

- Call a datum shift to the coordinates X = 0; Y = 0, etc., from a datum table.
- Execute the datum shift directly via cycle definition (see also page 8-42).
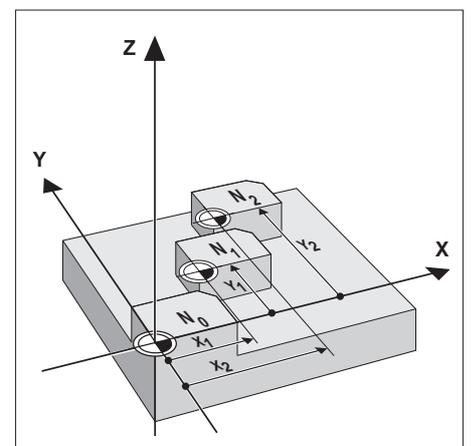
Fig. 8.49:   Only absolute datum shifts are possible from a datum table

8.5   Coordinate Transformations

Editing a datum table

Datum tables are edited in the PROGRAMMING AND EDITING mode:

SELECT TYPE + SHOW .D		Show the datum table.
↑ ↓ + SELECT		Select the desired file.
T A B + SELECT		Enter new file name, such as TAB.D .
Edit the datum table.		
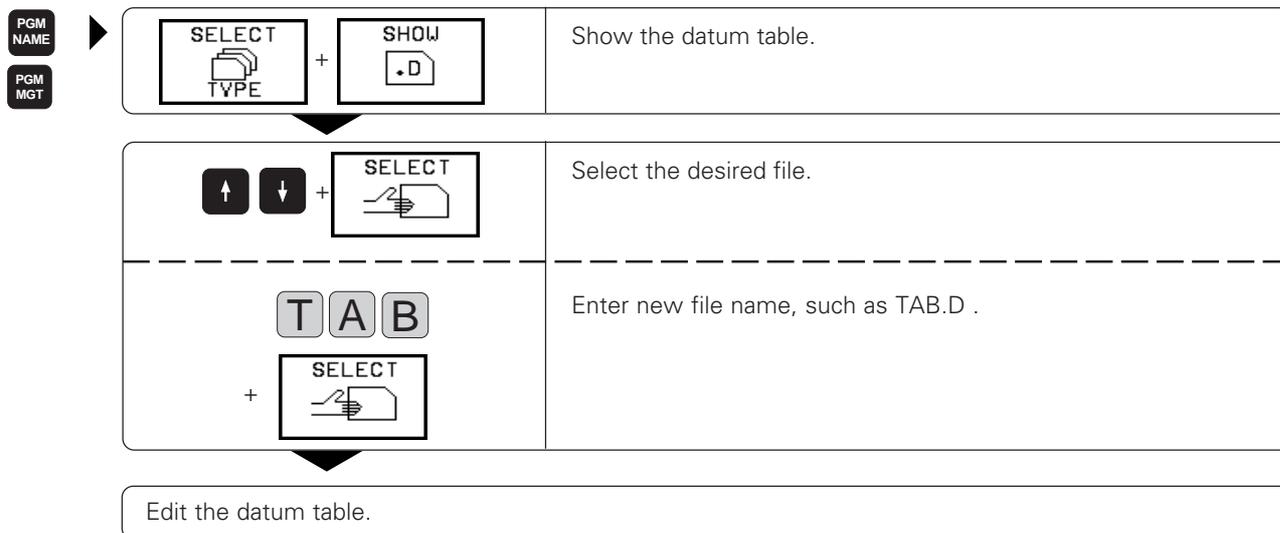
The soft keys comprise the following functions for editing:

BEGIN TABLE	END TABLE	PAGE ⇩	PAGE ⇧	INSERT LINE	DELETE LINE	NEXT LINE	

Function	Soft key
Go to the beginning of datum table	BEGIN TABLE
Go to the end of datum table	END TABLE
Page up	PAGE ⇧
Page down	PAGE ⇩
Insert line	INSERT LINE
Delete line	DELETE LINE
Enter line, go to beginning of next line	NEXT LINE

- If you are using only one datum table, be sure to activate the correct datum in the program run modes of operation.
- Datums from a datum table can be referenced either to the current datum or to the machine datum. The desired setting is made in MP 7475 (see page 11-14).
- New lines can only be inserted at the end of the file.

To leave a datum table:

PGM NAME ▶	SELECT TYPE SHOW .I	Select a different type of file, for example programs in ISO format.
PGM MGT		

↑ ↓ + SELECT	Choose the selected program.
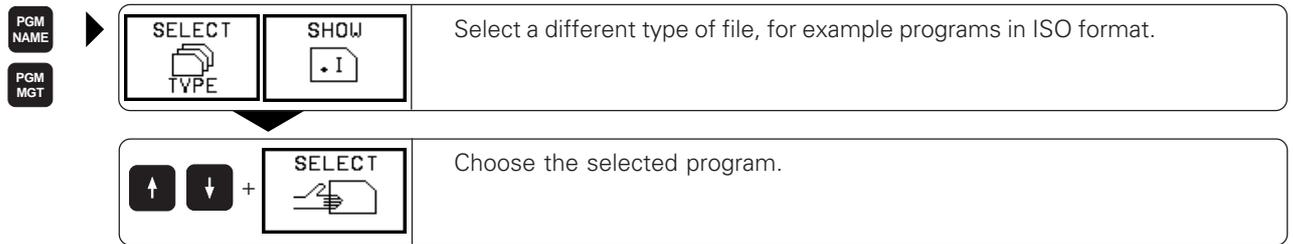
MIRROR IMAGE (G28)

Application

This cycle allows you to machine the mirror image of a contour in the machining plane.

Activation

The mirror image cycle becomes active immediately upon being defined. The mirrored axis is shown in the additional status display.

• If one axis is mirrored, the machining direction of the tool is reversed (except in fixed cycles).
• If two axes are mirrored, the machining direction remains the same.

The result depends on the location of the datum:

• If the datum is located on the contour to be mirrored, the part simply "flips over."
• If the datum is located outside the contour to be mirrored, the part also "jumps" to another location.

Input data

Enter the axes that you wish to mirror. The tool axis cannot be mirrored.

Cancellation

This cycle is cancelled by entering G28 without an axis.
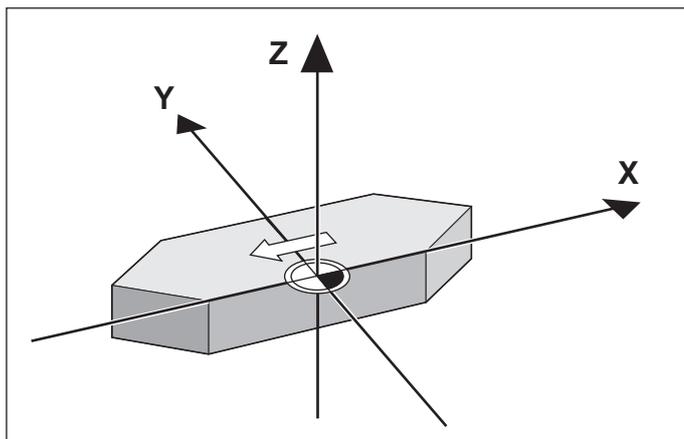
Fig. 8.50:   Mirroring a contour
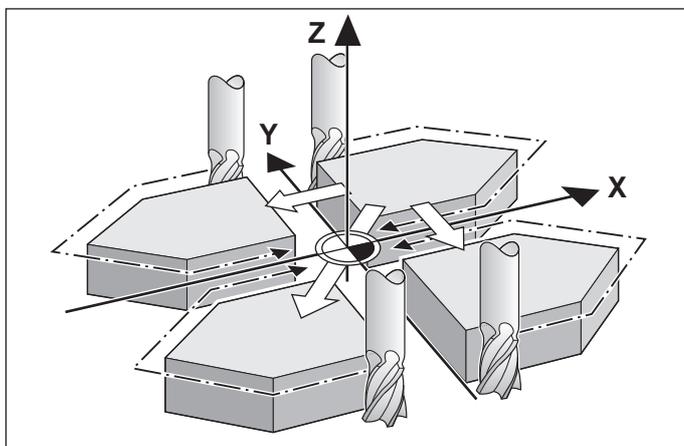
Fig. 8.51:   Repeated mirroring, machining direction
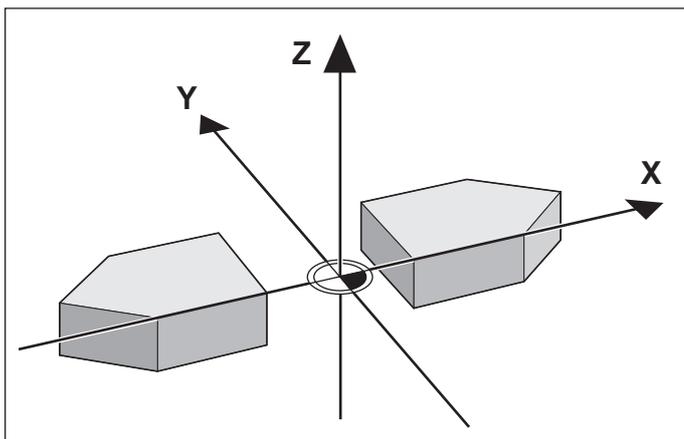
Fig. 8.52:   Datum located outside the contour to be mirrored

Example: Mirror image

A program section (subprogram 1) is to be exe-
cuted once as originally programmed at position
X+0/Y+0 ①, and then mirrored once in X ③ at
position X+70/Y+60 ②.
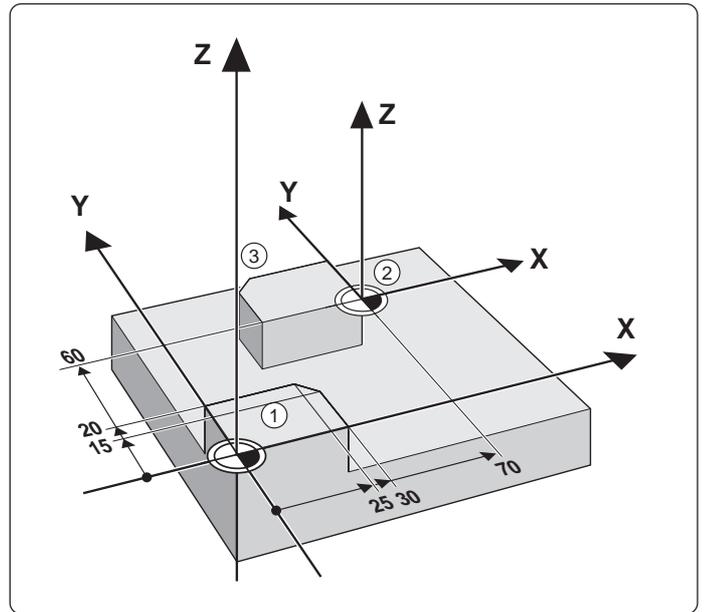
MIRROR IMAGE cycle in a part program

```
%S844I G71 * ............................................. Start of program
N10 G30 G17 X+0 Y+0 Z–20 * ................................... Define workpiece blank
N20 G31 X+100 Y+100 Z+0 *
N30 G99 T1 L+0 R+4 * ............................................. Define tool
N40 T1 G17 S1500 * .................................................. Call tool
N50 G00 G40 G90 Z+100 * ........................................ Retract in the infeed axis
N60 L1,0 * .................................................................. Version 1 unmirrored
N70 G54 X+70 Y+60 * ............................................... Shift datum
N80 G28 X * ............................................................... Activate mirroring
N90 L1,0 * .................................................................. Version 2, shifted and mirrored
N100 G28 * ................................................................ Cancel mirroring
N110 G54 X+0 Y+0 * ................................................ Cancel datum shift
N120 Z+100 M02 *

N130 G98 L1 *
 .
 .                                                            Same as subprogram on page 8-45
 .
N250 G98 L0 *
N99999 %S844I G71 *
```

ROTATION (G73)

Application

This cycle enables the coordinate system to be rotated about the active datum in the machining plane within a program.

Activation

Rotation becomes active as soon as it is defined. This cycle is also effective in the POSITIONING WITH MANUAL INPUT mode.

Reference axis for the rotation angle:

• X/Y plane   X axis
• Y/Z plane   Y axis
• Z/X plane   Z axis

The active rotation angle is displayed in the additional status display.

Input data

The rotation angle is entered in degrees (°).
Input range: –360° to +360° (absolute or incremental).

Cancellation

Rotation is cancelled by entering a rotation angle of 0°.

Example: Rotation

A contour (subprogram 1) is to be executed once as originally programmed referenced to the datum X+0/Y+0, and then rotated by 35° and referenced to the position X+70 Y+60.
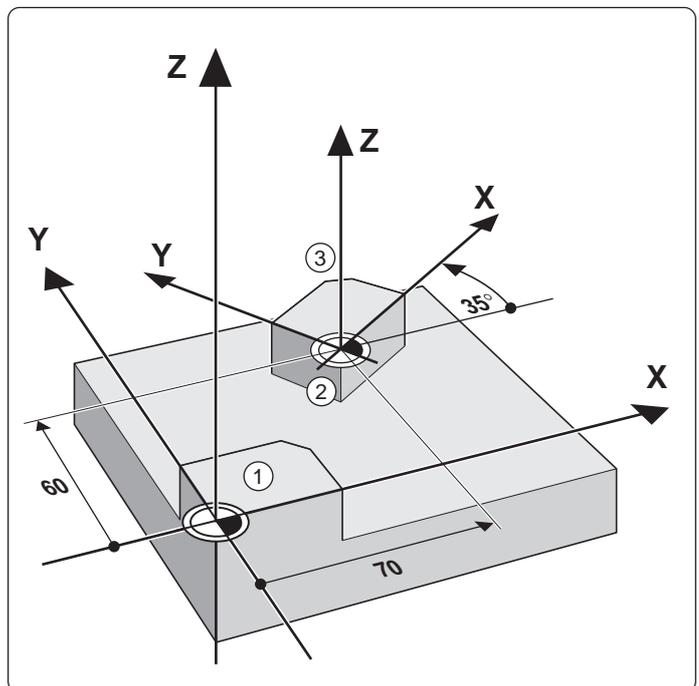
Continued on next page...

> **ROTATION cycle in a part program**
>
> %S846l G71 * ....Start of program
> N10 G30 G17 X+0 Y+0 Z–20 * ....Define workpiece blank
> N20 G31 X+100 Y+100 Z+0 *
> N30 G99 T1 L+0 R+4 * ....Define tool
> N40 T1 G17 S1500 * ....Call tool
> N50 G00 G40 G90 Z+100 * ....Retract in the infeed axis
> N60 L1,0 * ....Version 1 (not rotated)
> N70 G54 X+70 Y+60 *
> N80 G73 G90 H+35 *
> N90 L1,0 * ....Version 2 (shifted and rotated)
> N100 G73 G90 H+0 * ....Cancel rotation
> N110 G54 X+0 Y+0 * ....Cancel datum shift
> N120 Z+100 M02 *
> N130 G98 L1 *
> .
> .   Same as subprogram on page 8-45
> .
> N250 G98 L0 *
> N99999 %S846l G71 *

The corresponding subprogram (see page 8-45) is programmed after M2.

SCALING FACTOR (G72)

Application

G72 allows contours to be enlarged or reduced in size within a program, enabling you to program shrinkage and oversize allowances.

Activation

A scaling factor becomes effective as soon as it is defined.
The scaling factor can be applied

- in the machining plane, or on all three main axes at the same time (depending on MP 7410)
- to the dimensions in cycles
- to the parallel axes U, V, W

Input data

The cycle is defined by entering the factor *F*. The control then multiplies the coordinates and radii by *F* (as described under Activation above).
Enlargement: F > 1 (up to 99.999 999)
Reduction: F < 1 (down to 0.000 001)

Cancellation

To cancel the scaling factor, enter a scaling factor of 1 in the SCALING FACTOR cycle.

Prerequisite

It is advisable to set the datum to an edge or a corner of the contour before enlarging or reducing the contour.

Example: Scaling factor

A contour (subprogram 1) is to be executed as originally programmed at the manually set datum X+0/Y+0, and then referenced to position X+60/Y+70 and executed with a scaling factor of 0.8.
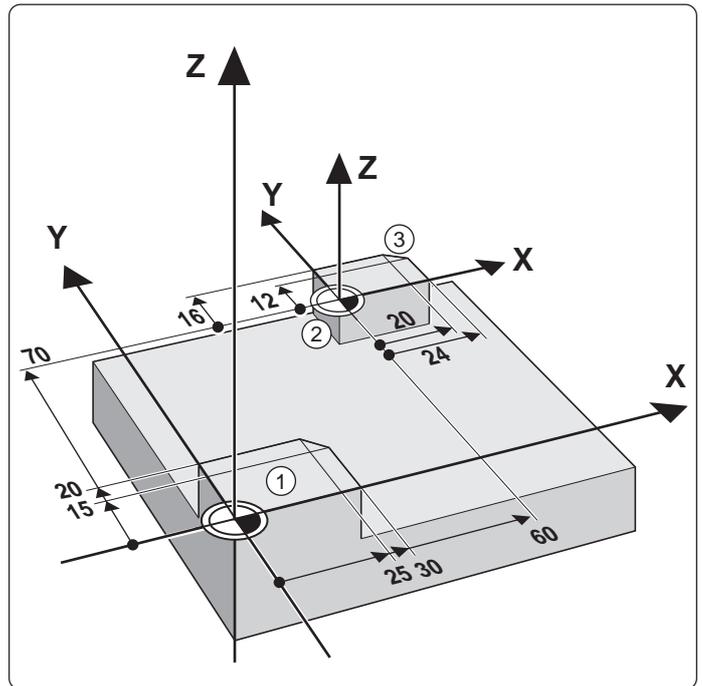
SCALING FACTOR cycle in a part program

```
%S847I G71 * ........................................................ Start of program
N10 G30 G17 X+0 Y+0 Z–20 * ................................. Define workpiece blank
N20 G31 X+100 Y+100 Z+0 *
N30 G99 T1 L+0 R+4 * ............................................. Define tool
N40 T1 G17 S1500 * .................................................. Call tool
N50 G00 G40 G90 Z+100 * ....................................... Retract in the infeed axis
N60 L1,0 * ................................................................ Version 1 (original size)
N70 G54 X+70 Y+60 *
N80 G72 F0,8 *
N90 L1,0 * ................................................................ Version 2 (shifted and reduced in size)
N100 G72 F1 * ......................................................... Cancel scaling factor
N110 G54 X+0 Y+0 * ................................................ Cancel datum shift
N120 Z+100 M02 *

N130 G98 L1 *
        .
        .                                          Same as subprogram on page 8-45
        .
N250 G98 L0 *
N99999 %S847I G71 *
```

The corresponding subprogram (see page 8-45) is programmed after M2.

8.6 Other Cycles

DWELL TIME (G04)

Application

This cycle causes the execution of the next block within a running program to be delayed by the programmed dwell time.

The dwell time cycle can be used for such purposes as chip breaking.

Activation

This cycle becomes effective as soon as it is defined. Modal conditions such as spindle rotation are not affected.

Input data

The dwell time is entered in seconds after G04 with F.
Input range: 0 to 30 000 sec. (approx. 8.3 hours) in increments of 0.001 sec.

Resulting NC block: N135 G04 F3*

PROGRAM CALL (G39)

Application and activation

Routines that are programmed by the user (such as special drilling cycles, curve milling or geometrical modules) can be written as main programs and then called like fixed cycles.

Input data

Enter the file name of the program to be called.

The program is called with

- G79 (separate block) or
- M99 (blockwise) or
- M89 (modally).

Example: Program call

A callable program (program 50) is to be called into a program via a cycle call.

Part program

.
.
.
G39 P01 50 .... "Program 50 is a cycle"
G00 G40 X+20 Y+50 M99 .... Call program 50
.
.
.

ORIENTED SPINDLE STOP (G36)

Application

The control can address the machine tool spindle as a 6th axis and rotate it to a given angular position. Oriented spindle stops are required for

- Tool changing systems with a defined tool change position
- Orientation of the transmitter/receiver window of the HEIDENHAIN TS 511 3D touch probe system

Activation

The angle of orientation defined in the cycle is positioned to by entering M19. If M19 is executed without a cycle definition, the machine tool spindle will be oriented to an angle which has been set in the machine parameters.
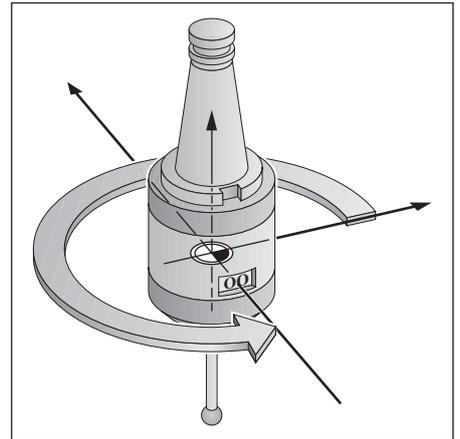
Fig. 8.53:   Oriented spindle stop

Apart from Cycle G36, oriented spindle stops can also be programmed in the machine parameters.

Prerequisite

The machine must first be set up for this cycle.

Input data

Angle of orientation S (according to the reference axis of the machining plane).

Input range: 0 to 360°
Input resolution: 0.1°

WORKING PLANE (G80)

> The functions for tilting the working plane are interfaced to the TNC and the machine tool by the machine manufacturer.
>
> **The following additional note applies to the TNC 426:**
> On some machines with swivel heads/tilting tables, the machine manufacturer determines whether the angles programmed in the cycle are interpreted as coordinates of the tilt axes or as solid angles. Your machine manual provides more detailed information on this subject.

The TNC supports machine tools with swivel heads (the tool is tilted) and/or tilting tables (the workpiece is tilted).

The program is written as usual in a main plane, such as the X/Y plane, but is executed in a plane that is tilted relative to the main plane.

Typical applications:

- Oblique holes
- Contours in an oblique plane

There are two ways to tilt the working plane:

- 3D ROT soft key in the MANUAL OPERATION and ELECTRONIC HANDWHEEL operation modes (see page 2-24)
- Cycle G80 WORKING PLANE in the part program

The tilting functions are coordinate transformations. The transformed tool axis (i.e., as calculated by the TNC) always remains parallel to the actual tool axis (the axis being positioned). The working plane is always perpendicular to the direction of the tool axis.

Fig. 8.54: Pre-position the tool perpendicular to the tilted plane

When tilting the working plane, the TNC differentiates between two machine types:

- Machines with tilting tables
- Machines with swivel heads

Machines with tilting tables:

- You must bring the **workpiece** into the desired position for machining by positioning the tilting table, for example with a G00 block.

- The position of the transformed tool axis **does not change** in relation to the machine-based coordinate system. Thus if you rotate the table—and therefore the workpiece—by 90° for example, the coordinate system **does not rotate**. If you press the Z+ axis direction button in the MANUAL OPERATION mode, the tool moves in Z+ direction.

- In calculating the transformed coordinate system the TNC considers only the mechanically influenced offsets of the particular tilting table (the so-called "translational" components).

Machines with swivel heads:

- You must bring the **tool** into the desired position for machining by positioning the swivel head, for example with a G00 block.

- The position of the transformed tool axis (like the position of the tool) **changes** in relation to the machine-based coordinate system. Thus if you rotate the swivel head—and therefore the tool—in the B axis by +90° for example, the coordinate system **rotates also**. If you press the Z+ axis direction button in the MANUAL OPERATION mode, the tool moves in X+ direction of the machine-based coordinate system.

- In calculating the transformed coordinate system the TNC considers the mechanically influenced offsets of the particular swivel head (the so-called "translational" components) **and** the offsets caused by tilting of the tool (3D tool length compensation).

Activation

Cycle G80 becomes effective immediately after definition in part programming. If the function TILTING PROGRAM RUN is set to ACTIVE in the MANUAL mode (see page 2-26), the angular value entered in this menu is overwritten by Cycle G80.

Input data

ROTARY AXIS AND TILT ANGLE:
The tilted axes of rotation together with the associated tilt angles.
The axes A, B and C are programmed using soft keys.

Entering an axis of rotation together with the associated tilt angle in Cycle G80 does not cause movement of the axis. The axis must be positioned to the required angle – for example with a G00 block.

Cancellation

To cancel Cycle G80, program G80 without entering an axis of rotation.

Pre-positioning

The following note applies to the TNC 426:
The machine manufacturer determines whether Cycle G80 positions the axes of rotation automatically or whether they must be pre-positioned in the part program. Your machine manual provides more detailed information.

If the axes are positioned automatically in Cycle G80:

• The TNC can only position controlled axes
• You can only use preset tools (with the full tool length defined in the G99 block or in the tool table)
• The position of the tool tip as referenced to the workpiece surface remains nearly unchanged after tilting
• The TNC tilts the working plane at the last programmed feed rate. The maximum feed rate that can be reached depends on the complexity of the swivel head or tilting table.

If the axes are not positioned automatically in Cycle G80, position them before defining the cycle, for example with a G00 block.

NC blocks

G00 G40 G90 Z+100*	
X+25 Y+10*	
G01 A+15 *F1000*	Pre-position in axis of rotation
G80 **A+15***	Define angle for compensating the slant
G00 G40 Z+80*	Activate compensation for Z axis
X–7.5 Y–10*	Activate compensation for X/Y axes

Position display in a tilted system

On activation of Cycle G80 the displayed positions (ACTL. and NOML.) and the datum indicated in the additional status display are referenced to the tilted coordinate system. The positions displayed immediately after cycle definition may not be the same as the coordinates of the last programmed position before Cycle G80.

As soon as you move an axis in a tilted system the compensation for this specific axis is activated. Move all axes to activate compensation for all axes.

Work space monitoring

The TNC only monitors the axes in the tilted system that are moved. If any of the software limit switches is traversed the TNC will display an error message.

Combining coordinate transformation cycles

When combining coordinate transformation cycles, always make sure the working plane is swiveled around the active datum. You can program a datum shift before activating Cycle G80. In this case, the machine-based coordinate system is shifted. If you program a datum shift after having activated Cycle G80, the tilted coordinate system is shifted. When resetting, it is important to use the reverse procedure—the cycle that was last defined should be reset first.

Example: 1. Activate datum shift
2. Activate tilting function
3. Activate rotation
.
.
Machining
.
.
1. Reset rotation
2. Reset tilting function
3. Reset datum shift

Automatic workpiece measurement in the tilted system

Cycle G55 (see page 7-23) enables you to have the TNC measure a workpiece in the tilted system automatically. The measured data are stored in Q parameters for further processing (for example, for printout).

8.6   Other Cycles

Procedure for working with Cycle G80 WORKING PLANE

1. Create program

- Define the tool (not required when TOOL.T is active).
- Call the tool.
- Retract the tool in the tool axis to a position where there is no danger of collision with the workpiece (clamping devices) during tilting.
- Position the tilt axis or axes with a G00 block to the appropriate angular value(s).
- Activate datum shift if required.
- Define Cycle G80 WORKING PLANE; enter the angular values for the tilt axes.
- Traverse all main axes (X, Y, Z) to activate compensation.
- Write the program as if the machining process were to be executed in a non-tilted plane.
- Reset Cycle G80 WORKING PLANE; program G80 without entering tilt axes.
- Reset datum shift if required.
- Pre-position the tilt axes to the 0° position if required.

2. Clamp workpiece

3. Preparations in the POSITIONING WITH MDI mode

Preposition the tilt axis/axes to the corresponding angular value(s). The angular value depends on the selected reference plane on the workpiece.

4. Preparations in the MANUAL OPERATION mode

Use the 3D-ROT soft key to set the function TILT WORKING PLANE to ACTIVE in the MANUAL OPERATION mode; enter the angular values for the tilt axes into the menu (see page 2-26).

The angular values entered in the menu must correspond to the actual position(s) of the tilted axis or axes, respectively. The TNC will otherwise calculate a wrong datum.

5. Set datum

- Manually by touching the workpiece with the tool in the non-tilted coordinate system (see page 2-7)
- Automatically by using a HEIDENHAIN 3D touch probe (see page 2-14)

6. Start part program in the PROGRAM RUN/FULL SEQUENCE mode

7. MANUAL OPERATION

Use the 3D-ROT soft key to set the function TILT WORKING PLANE to INACTIVE. Enter an angular value of 0° for each tilt axis into the menu (see page 2-26).

The TNC features two interfaces for data transfer between it and other devices.

Application examples:

• Downloading files into the TNC
• Transferring files from the TNC to external devices
• Printing files
• Remote operation of the TNC

The two interfaces can be used simultaneously.

9.1   Data Transfer with the TNC 426

The functions described in this chapter are operative in FE1 mode (see page 10-4). If you want to use these functions with a PC, you will need the TNC.EXE data transfer software from HEIDENHAIN.

To copy individual files into the TNC

➤ Press PGM MGT to call the file manager.
➤ Arrange the screen layout with the WINDOW soft key to show file names in both halves of the screen (see Chapter 1 under "File management on the TNC 426")
➤ In the left screen half, use the PATH soft key to select the directory into which you wish to copy the file from the external device.
➤ Move the highlight to the right screen half with the rightward arrow key.
➤ In the right screen half, use the PATH soft key to select the directory containing the files that you want to copy into the TNC.
➤ Move the highlight to the file you want to copy into the TNC.
➤ Press the COPY soft key and confirm with ENT. The TNC copies the highlighted file into the TNC.
➤ Close the file manager with END.

To copy multiple files into the TNC

➤ Press PGM MGT to call the file manager.
➤ Arrange the screen layout with the WINDOW soft key to show file names in both halves of the screen (see Chapter 1 "File management on the TNC 426").
➤ In the left screen half, use the PATH soft key to select the directory into which you wish to copy the file from the external device.
➤ Move the highlight to the right screen half with the rightward arrow key.
➤ In the right screen half, use the PATH soft key to select the directory containing the files that you want to copy into the TNC.
➤ Move the highlight to the first file you want to copy into the TNC.
➤ Shift the soft-key row.
➤ Select the file tagging function with the TAG soft key.
➤ Press the TAG FILE soft key. The TNC tags the highlighted file.
➤ Move the highlight to the next file you want to copy. Tag the file with TAG FILE. Tag all files you want to copy in this way.
➤ Press the COPY TAG soft key and confirm with ENT. Die TNC copies the tagged files into the TNC.
➤ Close the file manager with END.

Copying files out of the TNC

If you want to copy files out of the TNC, tag the files in the left half of the screen and copy them into the external device.

If you only want to copy a single file into or out of the TNC, use the COPY soft key after you have moved the highlight onto the file.

• If the TNC recognizes erroneously transferred program blocks, it will mark them with ERROR =.
  Correct these blocks in the PROGRAMMING AND EDITING mode.

• If you want to transfer files between two TNCs, start transmission from the receiving TNC.

• You can start a program in a program run mode of operation even if its transfer in the PROGRAMMING AND EDITING mode has not been completed yet. The program must not contain TOOL DEF blocks or label numbers.

• If you are transferring tables into the TNC you can overwrite individual lines or columns in the target table with the REPLACE FIELDS soft key. Prerequisites:
   – The target table must exist.
   – The file to be transferred must only contain the columns (or lines) you want to replace.

9.2   Data Transfer with the TNC 425, TNC 415 B and TNC 407

To select external data transfer:

EXT	Menu for external data transfer appears on the screen.

The screen is divided into two halves:

Active interface
(RS-232 or RS-422)

Interface mode
(FE1, FE2, ME, EXT1, EXT2);
indicated file type

```
MANUAL        PROGRAMMING  AND  EDITING
OPERATION

TNC:                             RS232/FE1:
 FILE NAME        BYTE  STATUS     FILE NAME        SECTORS STATUS
79153        .H   1100           $MDI        .H    1
FRESADOR     .H   462            1           .H    1
TAB1         .T   1470           10          .H    1
TOOL         .T   1470    M      1111        .H    1
ERFG         .I   74             115         .H    1
PAL1         .P   756            123         .H    1
PAL2         .P   756            123456      .H    1
DD25LP       .D   462            200         .H    2
SK50         .D   462            22742602    .H    9
1            .A   218            300         .H    1
11V          .A   696            3500        .H    1
2            .A   190            3501        .H    1
57  FILE(S) 150016 BYTES VACANT  46  FILE(S) 680 SECTORS VACANTT
  PAGE     PAGE    TRANSFER  TRANSFER  TRANSFER  SELECT   WINDOW
   ⇧        ⇩     TNC⇒EXT   TNC → EXT  TNC ? EXT  TYPE            END
```

Files in
the TNC

Files (if any) in external
storage device

☞ If you select the data transfer function from a tool table or pocket table, only the functions

TRANSFER
TNC⇒EXT   and   TRANSFER
TNC⇐EXT

are available.

9.2   Data Transfer with the TNC 425, TNC 415 B and TNC 407

Selecting and transferring files

The data transfer functions are provided in a soft-key row.

Soft-key row in the PROGRAMMING AND EDITING mode of operation:
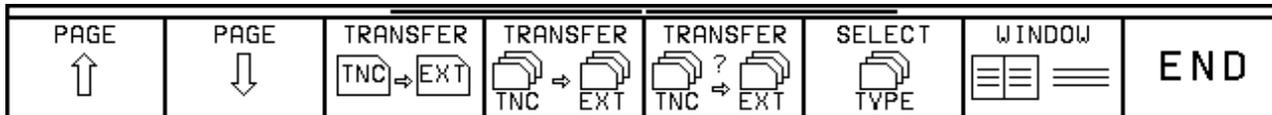
Selecting files

Use the arrow keys to select the desired file. The PAGE soft keys are for scrolling up and down in the file directory. The SELECT TYPE soft key has the same function as described earlier (see Chapter 1 "File management on the TNC 425, TNC 415 B and TNC 407").

Transferring files from the TNC to an external device

The highlight is on a file that is stored in the TNC.

Function	Soft key
Transfer selected file	TRANSFER TNC⇒EXT
Transfer all files	TRANSFER TNC ⇒ EXT
Select files consecutively for individual transfer. Press ENT to transfer, otherwise press NO ENT	TRANSFER TNC ? ⇒ EXT

Selecting the file type

Use the SELECT TYPE soft key to select other types of files.

Select screen layout

Use the WINDOW soft key to select or deselect a split screen. The single screen mode can be selected both for TNC files and for externally stored files. If the highlight is at left, only TNC files are displayed. If the highlight is at right, only externally stored files are displayed.

Transferring files from an external device to the TNC

Use the cursor key to move the highlight to a file that is stored in the external device.

Function	Soft key
Transfer selected file	TRANSFER [TNC] ⇐ [EXT]
Transfer all files	TRANSFER TNC ⇐ EXT
Select files consecutively for individual transfer. Press ENT to transfer, otherwise press NO ENT	TRANSFER ? TNC ⇐ EXT

Interrupt transfer

You can interrupt data transfer by pressing the END key or the END soft key.

> • If the TNC recognizes erroneously transferred program blocks, it will mark them with ERROR =.
>   These blocks must then be corrected in the PROGRAMMING AND EDITING mode.
>
> • If you want to transfer files between two TNCs, start transmission from the receiving TNC.

Blockwise transfer

The menu to the right is for blockwise transfer (see page 3-11). First select as usual the name of the file to be transferred blockwise. Then start data transfer with the SELECT soft key.

Fig. 9.1:   Menu for blockwise transfer

> When aborting blockwise transfer, you may have to reset the interface with the CLOSE RS-232-C soft key.

9.3   Pin Layout and Connecting Cable for the Data Interfaces

RS-232-C/V.24 Interface

HEIDENHAIN devices
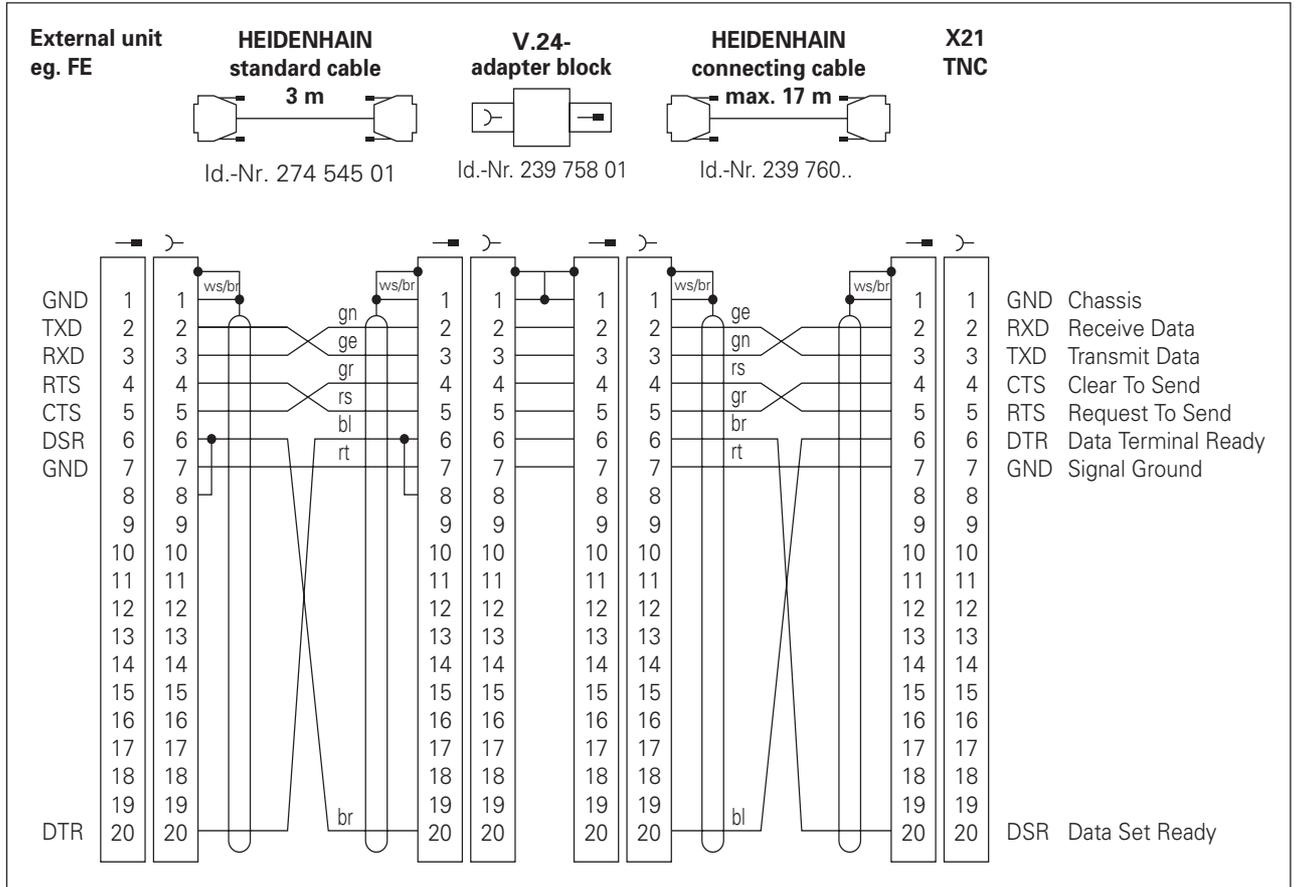
Fig. 9.2:   Pin layout of the RS-232-C/V.24 interface for HEIDENHAIN devices

The connector pin layout on the adapter block differs from that on the TNC logic unit (X21).

9.3   Pin Layout and Connecting Cable for the Data Interfaces

Non-HEIDENHAIN devices

The connector pin layout on a non-HEIDENHAIN device may differ considerably from that on a HEIDENHAIN device, and depends on the unit and the type of data transfer. Fig. 9.3 shows the connector pin layout on the adapter block.
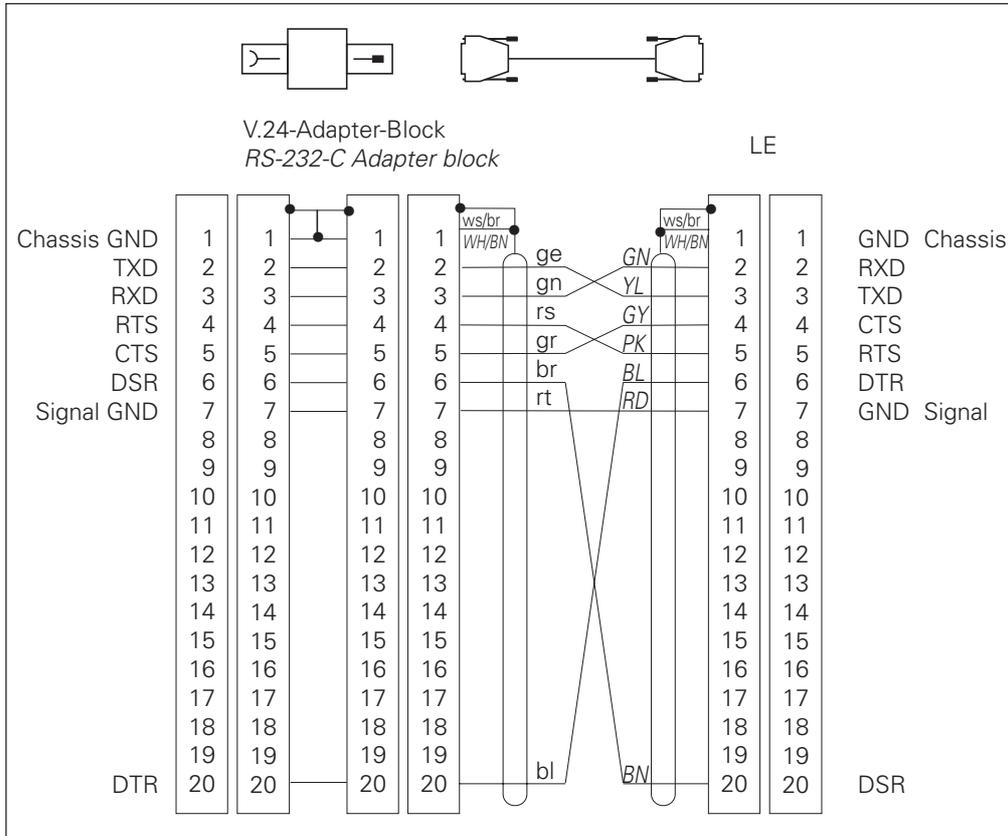
Fig. 9.3:   Connecting a non-HEIDENHAIN device to the RS-232-C/V.24 interface

9.3   Pin Layout and Connecting Cable for the Data Interfaces

RS-422/V.11 Interface

Only non-HEIDENHAIN devices are connected to the RS-422 interface.
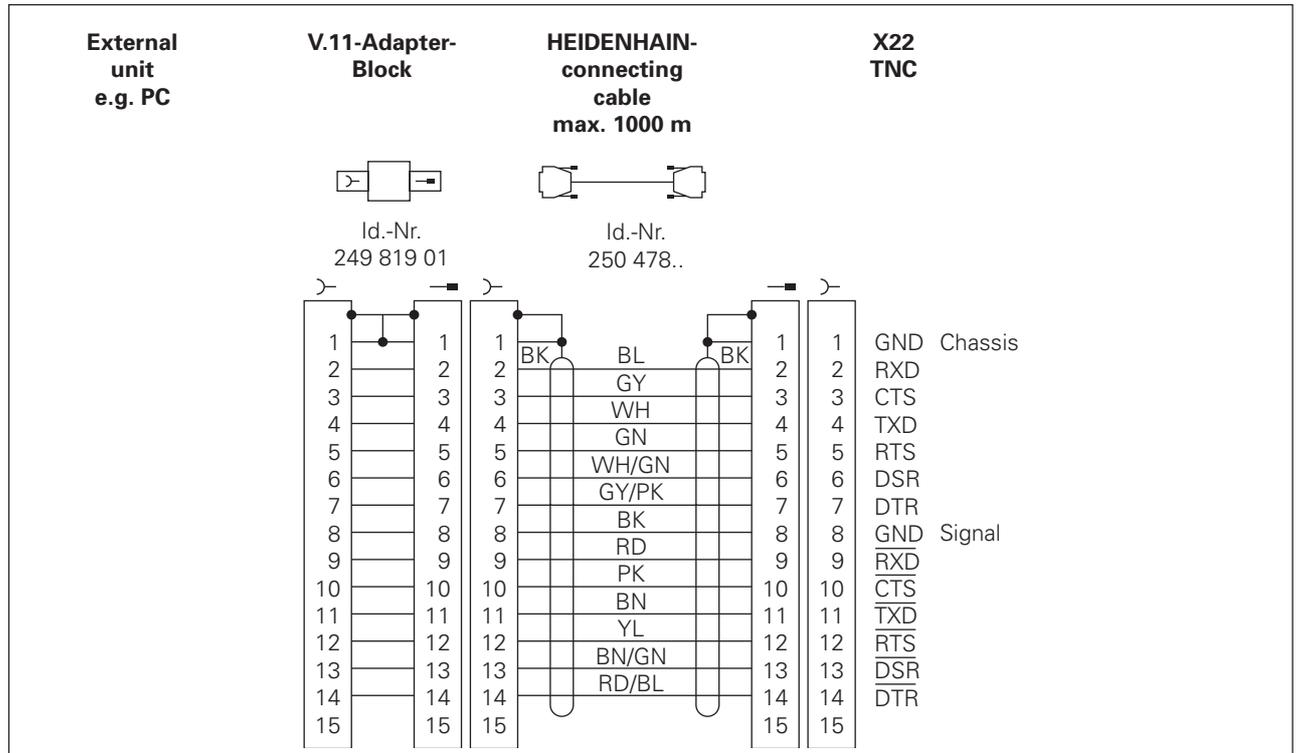
Fig. 9.4:   Pin layout of the RS-422/V.11 interface

The pin layouts on the TNC logic unit (X22) and on the adapter block are identical.

9.4   Preparing the Devices for Data Transfer

HEIDENHAIN devices

HEIDENHAIN devices (FE floppy disk unit and ME magnetic tape unit) are already adapted to the TNC. They can be used for data transfer without further adjustments.

Example: FE 401 floppy disk unit

- Connect the power cable to the FE
- Connect the FE and TNC with the data interface cable
- Switch on the FE
- Insert a diskette into the upper drive
- Format the diskette if necessary
- Set the interface (see page 10-4)
- Transfer the data

- The memory capacity of a floppy disk is given in sectors.

- The baud rate can be set at the FE 401.

Non-HEIDENHAIN devices

The TNC and non-HEIDENHAIN devices must be adapted to each other.

To adapt a non-HEIDENHAIN device to the TNC

- PC: Adapt the software
- Printer: set the DIP switches

To adapt the TNC to a non-HEIDENHAIN device

Set the user parameters:

- 5020.0 to 5210.0 for EXT1
- 5020.1 to 5210.1 for EXT2

The two settings can be adjusted, for example, to a PC (e.g. EXT1) or to a printer (EXT2).

The MOD functions provide additional displays and input possibilities. The available MOD functions depend on the selected operating mode.

Functions and displays available in the PROGRAMMING AND EDITING mode of operation:

- Display NC software number
- Display PLC software number
- Enter code number
- Set data interface
- Machine-specific user parameters
- HELP files (if provided)

Fig. 10.1:   MOD functions in the PROGRAMMING AND EDITING mode

In the TEST RUN mode of operation:

- Display NC software number
- Display PLC software number
- Enter code number
- Set data interface
- Graphic display of the workpiece blank in the working area of the machine
- Machine-specific user parameters
- HELP files (if provided)

Fig. 10.2:   MOD functions in the TEST RUN mode

In all other modes:

- Display NC software number
- Display PLC software number
- Display code digits for installed options
- Select position display
- Unit of measurement (mm or inch)
- Programming language (HEIDENHAIN or ISO)
- Selecting the axes for generating L blocks *(conversational programming only)*
- Axis traverse limits
- Display datums
- Display machining time
- HELP files (if provided)

Fig. 10.3:   MOD functions in a machine operating mode

10.1 Selecting, Changing and Exiting the MOD Functions

To select the MOD functions:

If necessary ○	Change to the desired mode of operation.
MOD	Select MOD functions.

To change the MOD functions:

Use the arrow keys to move the highlight to the desired MOD function.	
ENT Repeatedly	Page through the MOD functions until you find the desired function.
e.g. **5** ENT	Enter the appropriate numbers and confirm entry with ENT.

To exit the MOD functions:

E N D or END	Close the MOD functions.

10.2 Software Numbers and Option Numbers

The software numbers of the NC and PLC are displayed in the MOD function opening screen. Directly below them are the code numbers for the installed options (only for conversational programming).

- Digitizing option   OPT: 1
- Digitizing and measuring touch probe options   OPT: 11

10.3 Code Numbers

A code number is required for access to certain functions:

Function	Code number
To cancel file erase and edit protection (status P)	86 357
To select user parameters	123

10.4 Setting the External Data Interfaces

Press the soft key marked RS 232- / RS 422 - SETUP to call a menu for setting the external data interfaces.

- MODE OF OP. – Type of external storage device: FE1, FE2, ME, EXT1, EXT2, LSV2
- BAUD RATE – Sets the data transfer speed (110 to 38400 baud)
- ASSIGN – Assigns either the RS-232 or the RS-422 interface to the operating modes
- PRINT – Outputs digitized data through RS-232, RS-422 or FILE

Setting the RS-232 interface

The mode of operation and baud rates for the RS-232 interface are entered in the upper left of the screen.

Setting the RS-422 interface

The mode of operation and baud rates for the RS-422 interface are entered in the upper right of the screen.

Selecting the OPERATING MODE

External device	OPERATING MODE
HEIDENHAIN floppy disk units	
• FE 401 B	FE1
• FE 401 with program no. 230 626 03	
or higher	FE1
HEIDENHAIN FE 401 floppy disk unit with program number below 230 626 02	FE2
PC with HEIDENHAIN data transfer software TNC. EXE	FE2
HEIDENHAIN ME 101 magnetic tape unit (no longer produced)	ME **426**
Non-HEIDENHAIN devices such as printers, tape punchers, PCs without TNC. EXE	EXT1, EXT2
PC with HEIDENHAIN software TNC REMOTE for remote operation	LSV2

- The HEIDENHAIN ME 101 magnetic tape unit (ME mode of operation) can only be used in the TNC mode of operation PROGRAMMING AND EDITING (not on TNC 426).

The following note applies to the TNC 426:
- The functions "Transfer all files", "Transfer selected file" and "Transfer directory" are not available in the operating modes FE2 and EXT.

Setting the BAUD RATE

On the TNC 425, TNC 415 B and TNC 407 controls, the baud rate (data transfer speed) can be selected from 110 to 38 400 baud. The baud rate of the ME 101 is 2400 baud.

On the TNC 426, the baud rate (data transfer speed) can be selected from 110 to 115 200 baud.

10.4 Setting the External Data Interfaces

ASSIGN

This function sets the destination for the transferred data.

Applications:

- Transferring values with Q parameter function D15
- Transferring values with Q parameter function D16 (only with TNC 426)
- Path on the TNC's hard disk in which the digitized data are stored (only with TNC 426)

The TNC mode of operation determines whether the PRINT or PRINT-TEST function is used:

TNC mode of operation	Transfer function
PROGRAM RUN, SINGLE BLOCK	PRINT
PROGRAM RUN, FULL SEQUENCE	PRINT
TEST RUN	PRINT-TEST

You can set PRINT and PRINT-TEST as follows:

Function	Path to be entered (TNC 426)	Setting (TNC 407/TNC 415 B/TNC 425)
Transfer data with D15/D16 via RS-232	RS232:\...........	RS-232
Transfer data with D15/D16 via RS-422	RS422:\...........	RS-422
Save data to the TNC's hard disk or in the TNC memory	TNC:\...........	FILE
Save data in directory in which the program with D15/D16 is located (TNC 426). Do not save data (TNC 407/TNC 415 B/TNC 425)	- vacant -	- vacant -

File names

Data	Mode of operation	File name
Values with D15	PROGRAM RUN	%D15RUN.A
Values with D15	TEST RUN	%D15SIM.A
Values with D16	PROGRAM RUN	%D16RUN.A
Values with D16	TEST RUN	%D16SIM.A

10.5 Machine-Specific User Parameters

The machine tool builder can assign functions to up to 16 user parameters. For more detailed information on user parameters, refer to your machine operating manual.

10.6 Showing the Workpiece in the Working Space

The DATUM SET soft key enables you to graphically check the position of the workpiece blank in the machine's working space and to activate work space monitoring in the TEST RUN mode of operation.

Available traversing range/datums, referenced to the displayed workpiece blank

Size of the blank

Coordinate system

Working space

Workpiece blank with orthographic projections

Overview of functions

Function	Soft key
Move workpiece blank to the left (graphically)	← ⊕
Move workpiece blank to the right (graphically)	→ ⊕
Move workpiece blank forward (graphically)	↙ ⊕
Move workpiece blank backward (graphically)	↗ ⊕
Move workpiece blank upward (graphically)	↑ ⊕
Move workpiece blank downward (graphically)	↓ ⊕
Show workpiece blank referenced to the set datum	⊔
Shift the soft-key row	▷ or ◁
Show the entire traversing range referenced to the workpiece blank	⊢⟶⊣
Show the machine datum in the working space	M91 ⊕
Show a position determined by the machine tool builder (e.g. tool change position) in the working space	M92 ⊕
Show the workpiece datum in the working space	⊕
Disable (OFF) or enable (ON) work space monitoring during test run	⊢⟶⊣ OFF/ON
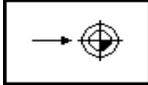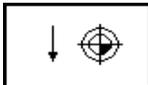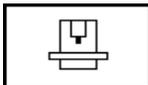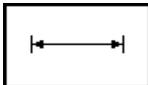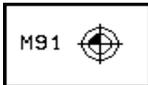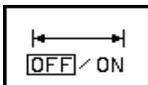
10.7 Position Display Types

The positions indicated in figure 10.4 are:

- Starting position Ⓐ
- Target position of the tool Ⓩ
- Workpiece datum Ⓦ
- Scale reference point Ⓜ
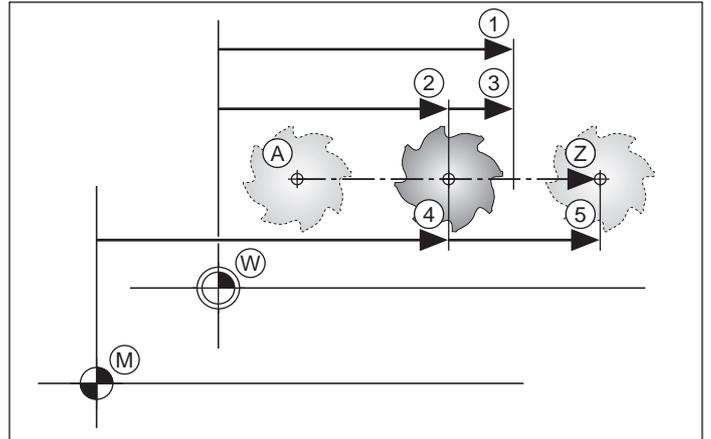
Fig. 10.4:   Characteristic positions on the workpiece and scale

The TNC position display can show the following coordinates:

- Nominal position: the value presently commanded by the TNC ① .... NOML.

- Actual position: the position at which the tool is presently located ② .... ACTL.

- Servo lag: the difference between nominal and actual positions ③ .... LAG

- Reference position: the actual position as referenced to the scale reference point ④ .... REF

- Distance remaining to the programmed position: the difference between actual and target positions ⑤ .... DIST.

With the MOD function POSITION DISPLAY 1 (see figure 10.3) you can select the position display in the status display.

With the MOD function POSITION DISPLAY 2 (see figure 10.3) you can select the position display in the additional status display.

10.8 Unit of Measurement

This MOD function determines whether coordinates are displayed in millimeters (metric system) or inches.

- To select the metric system (e.g., X = 15.789 mm),
  set the CHANGE MM/INCH function to MM.
  The value is displayed with 3 digits after the decimal point.
- To select the inch system (e.g., X = 0.6216 inch),
  set the CHANGE MM/INCH function to INCH.
  The value is displayed to 4 decimal places.

10.9 Programming Language for $MDI

The PROGRAM INPUT mod function lets you decide whether to program the $MDI file in HEIDENHAIN conversational dialog or in G-codes in accordance with ISO.

- To program the $MDI.H file in conversational dialog,
  set the PROGRAM INPUT function to HEIDENHAIN.
- To program the $MDI.I file according to ISO,
  set the PROGRAM INPUT function to ISO.

10.10 Selecting the Axes for Generating L Blocks (conversational programming only)

The AXIS SELECTION input field enables you to define the current tool position coordinates that are transferred to an L block. To generate a separate L block, press the "ACTUAL POSITION CAPTURE" soft key (see User's Manual for HEIDENHAIN Conversational Programming). On the TNC 407, up to 3 coordinates can be transferred; on the TNC 415 B, TNC 425 and TNC 426 controls, you can transfer up to 5 coordinates. The axes are selected by bit-oriented definition, similar to programming the machine parameters.

```
AXIS SELECTION      %11111      Transfer the X, Y, Z, IV, and V axes
AXIS SELECTION      %01111      Transfer the X, Y, Z, and IV axes
AXIS SELECTION      %00111      Transfer the X, Y, and Z axes
AXIS SELECTION      %00011      Transfer the X and Y axes
AXIS SELECTION      %00001      Transfer the X axis
```

10.11 Axis Traverse Limits

The AXIS LIMIT mod function allows you to set limits to axis traverse within the machine's actual working envelope.

Possible application:
to protect an indexing fixture against tool collision.

The maximum range of traverse of the machine tool is defined by software limit switches. This range can be additionally limited through the AXIS LIMIT mod function. With this function you can enter the maximum and minimum traverse positions for each axis, referenced to the machine datum.
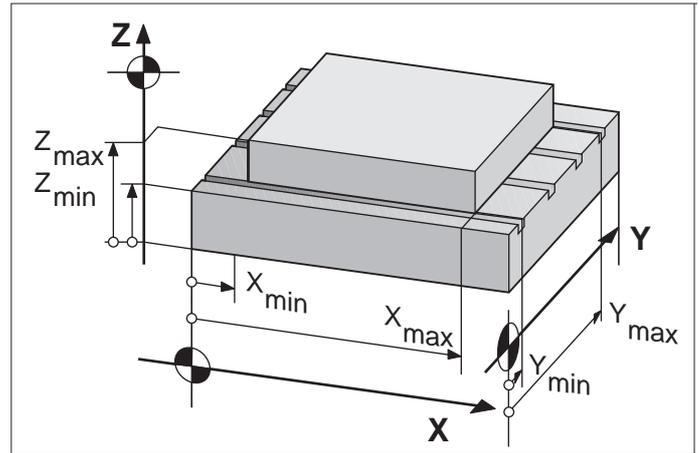
Fig. 10.5:   Orienting traverse limits to workpiece size

Working without additional traverse limits

To allow a machine axis to use its full range of traverse, enter the maximum traverse of the TNC (+/– 99999.999 mm) as the AXIS LIMIT.

To find and enter the maximum traverse:

Set the POSITION DISPLAY mod function to REF.	
Move the spindle to the positive and negative end positions of the X, Y and Z axes.	
Write down the values, including the algebraic sign.	
MOD	Select the MOD functions.
AXIS LIMIT	Enter the values that you wrote down as LIMITS in the corresponding axes.
END	Exit the MOD functions.

- The tool radius is not automatically compensated in the axis traverse limit values.
- The traverse range limits and software limit switches become active as soon as the reference points are passed over.

Datum display

The values shown at the lower left of the screen are the manually set datums referenced to the machine datum. They cannot be changed in the menu.

10.12 HELP files

Help files are a way to find information quickly that you would otherwise have to search for in a manual. Help files can aid you in situations in which you need clear instructions before you can continue (for example, to retract the tool after an interruption in power). The miscellaneous functions may also be explained in a help file.

Help files are not provided on every machine. Your machine tool builder can provide you with further information on this feature.

To call help files:

MOD	Select the MOD functions.
HELP	Call the last active HELP file.
If desired **PGM NAME**	Call other HELP files.
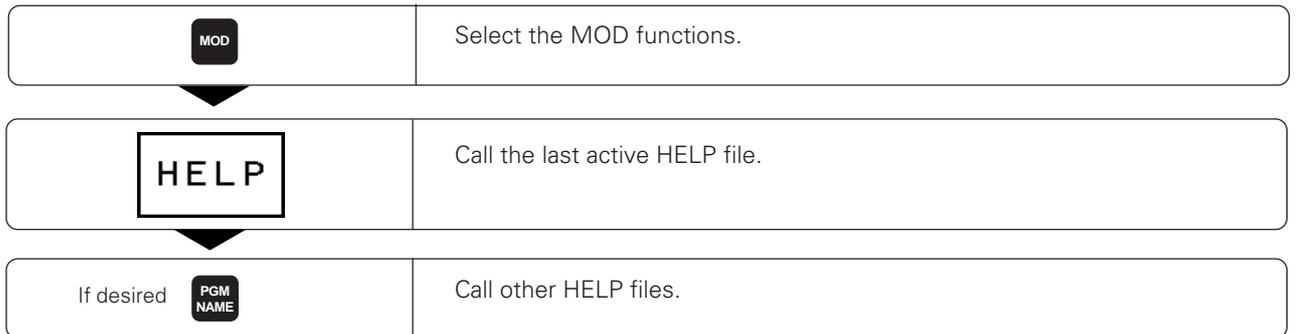
```
PROGRAMMING  AND  EDITING              PROGRAMMING
                                       AND EDITING

FILE: TOOLCHANGER      LINE: 1    COLUMN: 1    INSERT
COMMANDS  FOR  THE  TOOL  CHANGER  !!!


#0001  CHAIN  FORWARD
#0002  CHAIN  BACKWARD
[END]




ACTL.      X     +25,3684     Y    -250,3600
           Z     -25,0000     B    +331,0000
           C     +12,5000
T                            F  0          M  5/9

INSERT    MOVE    MOVE    PAGE    PAGE    BEGIN   END     FIND
OVERWRITE WORD    WORD     ⇩       ⇧     TEXT    TEXT
          >>      <<
```

Fig. 10.6 HELP file in a machine operating mode

11.1 General User Parameters

General user parameters are machine parameters affecting TNC settings that the user may want to change in accordance with his requirements. Some examples of user parameters are:

- Dialog language
- Interface behavior
- Traversing speeds
- Sequence of machining
- Effect of overrides

Input possibilities for machine parameters

Machine parameters can be programmed as
- Decimal numbers:
  Enter only the number.
- Pure binary numbers:
  Enter a percent sign (%) before the number.
- Hexadecimal numbers:
  Enter a dollar sign ($) before the number.

Example:

Instead of the decimal number 27 you can enter the binary number % 11011 or the hexadecimal number $1B.

The individual machine parameters can be entered in the different number systems.

Some machine parameters have more than one function. The input value for these machine parameters is the sum of the individual values. For these machine parameters the individual values are preceded by a plus sign (*+0*, *+1*, etc.)

Selecting general user parameters

General users parameters are selected with code number 123 in the MOD functions.

> The MOD functions also include machine specific user parameters (USER PARAMETERS).

External data transfer

Integrating TNC interfaces EXT1 (5020.0) and EXT2 (5020.1) to an external device:

MP5020.x   7 data bits (ASCII code, 8th bit = parity): **+0**
8 data bits (ASCII code, 9th bit = parity): **+1**

Block Check Character (BCC) any BCC:**+0**
Block Check Character (BCC) BCC control character not permitted: **+2**

Transmission stop through RTS active: **+4**
Transmission stop through RTS inactive: **+0**

Transmission stop through DC3 active: **+8**
Transmission stop through DC3 inactive: **+0**

Character parity even: **+0**
Character parity odd: **+16**

Character parity not desired: **+0**
Character parity desired: **+32**

$1^1/_2$ Stop bits: **+0**
2 Stop bits: **+64**
1 Stop bit: **+128**
1 Stop bit: **+192**

Example:
Use the following setting to adjust the TNC interface EXT2 (MP 5020.1) to an external non-HEIDENHAIN device:
8 data bits, any BCC, transmission stop through DC3, even character parity, character parity desired, 2 stop bits
Input value: 1+0+8+0+32+64 = 105 (entry value for MP 5020.1)

Interface type for EXT1 (5030.0) and EXT2 (5030.1):

MP5030.x   Standard transmission: **0**
Interface for blockwise transfer: **1**

Define data transfer through EXT1 (xxxx.0) and EXT2 (xxxx.1):

<div style="text-align:right">426</div>

MP5200.x	Start transmission (STX):	*Decimal value for ASCII character (0 to 127)*
MP5201.x	End transmission (ETX)	
MP5202.x	Data input (1st character) H	
MP5203.x	Data input (2nd character) E	
MP5204.x	Data output (1st character) H	
MP5205.x	Data output (2nd character) A	
MP5206.x	Start of heading (SOH)	
MP5207.x	End of transmission block (ETB)	
MP5208.x	Positive acknowledgement (ACK)	
MP5209.x	Negative acknowledgement (NAK)	
MP5210.x	End of transmission (EOT)	

3D touch probes and digitizing

Select touch probe

MP6200   Triggering touch probe: **0**
Measuring touch probe: **1**

Select signal transmission

MP6010   Touch probe with cable transmission: **0**
Touch probe with infrared transmission: **1**

Probing feed rate for triggering touch probes

MP6120   **80 to 3 000 [mm/min]**

Maximum traverse to first probe point

MP6130   **0 to 99 999.9999 [mm]**

Safety clearance to probing point during automatic measurement

MP6140   **0 to 99 999.9999 [mm]**

Rapid traverse for triggering touch probes

MP6150   **1 to 300 000 [mm/min]**

Measure center misalignment of the stylus when calibrating a triggering touch probe

MP6160   No 180° rotation of the 3D touch probe during calibration: **0**
M function for 180° rotation of the 3D touch probe during calibration: **1 to 88**

Lubricating the touch probe axis during digitizing with a triggering touch probe   426

MP6220   Traverse in the touch probe axis at the end of line: **0 to 99 999.9999 [mm]**

Lubricating the touch probe axis during digitizing with a triggering touch probe   426

MP6221   Time after which the axis should be lubricated: **0 to 65 535 [min]**

3D touch probes and digitizing

MP6300   **Reserved**

Infeed of the stylus when digitizing with a measuring touch probe

MP6310   **0.1 to 2.0000 [mm]** (recommended input value: 1mm)

Measure center misalignment of the stylus when calibrating a measuring touch probe

MP6321   Measure center misalignment: **0**
                Do not measure center misalignment: **1**

Assign touch probe axis to machine axis for a measuring touch probe

☞ Ensure that the touch probe axes are correctly assigned to the machine axes. Wrong assignment could lead to a stylus break.

MP6322.0   Machine X axis parallel to
                Touch probe X axis: **0**
                Touch probe Y axis: **1**
                Touch probe Z axis: **2**
MP6322.1   Machine Y axis parallel to
                Touch probe X axis: **0**
                Touch probe Y axis: **1**
                Touch probe Z axis: **2**
MP6322.2   Machine Z axis parallel to
                Touch probe X axis: **0**
                Touch probe Y axis: **1**
                Touch probe Z axis: **2**

Maximum stylus deflection on measuring touch probes

MP6330   **0.1 to 4.0000 [mm]**

Feed rate for positioning measuring touch probes at MIN point and approaching contour

MP6350   **10 to 3 000 [mm/min]**

Probing feed rate for measuring touch probes

MP6360   **10 to 3 000 [mm/min]**

Rapid traverse for measuring touch probes in the probe cycle

MP6361   **10 to 3 000 [mm/min]**

Feed rate decrease when the stylus of a measuring touch probe is deflected to the side

                The TNC decreases the feed rate according to a preset characteristic curve. The minimum input value is 10% of the programmed digitizing feed rate.
MP6362   Feed rate decrease not active: **0**
                Feed rate decrease active: **1**

Target window for digitizing contour lines with measuring touch probes

                When you are digitizing contour lines the individual contour lines do not end exactly in their starting points. With machine parameter MP 6390 you can define a square target window within which the end point must lie after the touch probe has orbited the model. Enter half the side length of the target window for the input value.

MP6390   **0.1 to 4.0000 [mm]**

3D touch probes and digitizing

Radius measurement with the TT 120 touch probe: Probing direction

MP6505   Positive probing direction in the angle reference axis (0° axis): **0**
Positive probing direction in the +90° axis: **1**
Negative probing direction in the angle reference axis (0° axis): **2**
Negative probing direction in the +90° axis: **3**

Probing feed rate for second measurement with TT 120, stylus shape, corrections in TOOL.T

MP6507   Calculate feed rate for second measurement with TT 120, with constant tolerance: **+0**
Calculate feed rate for second measurement with TT 120, with variable tolerance: **+1**
Constant feed rate for second measurement with TT 120: **+2**

TT 120 with cylindrical stylus: **+0**
TT 120 with cubic stylus: **+64**

Automatically enter the results in TOOL.T: **+0**
Do not correct TOOL.T: **+128**

Maximum permissible error of measurement for measuring rotating tools with the TT 120

Required for calculating the probing feed rate in connection with MP6570.

MP6510   **0.001 to 0.999 [mm]** (recommended input value: 0.005 mm)

Feed rate for probing a tool with the TT 120 during standstill of the tool

MP6520   **10 to 3 000 [mm/min]**

Radius measurement with the TT 120: Distance from lower edge of tool to upper edge of stylus

The value entered for MP6530 is added to the value defined in the tool table for TT:L-OFFS.

MP6530   **0.0001 to 9 999.9999 [mm]**

Rapid traverse for the TT 120 touch probe in the probe cycle

MP6550   **10 to 10 000 [mm/min]**

Measuring rotating tools: Permissible rotational speed at the circumference of the milling tool

MP6570   **40.000 to 120.0000 [m/min]**

Stylus center REF coordinates of the TT 120 touch probe

MP6580.0   X axis: **–99 999.9999 to 99 999.9999 [mm]**

MP6580.1   Y axis: **–99 999.9999 to 99 999.9999 [mm]**

MP6580.2   Z axis: **–99 999.9999 to 99 999.9999 [mm]**

TNC displays, TNC editor

Programming station

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

Acknowledgment of POWER INTERRUPTED message after switch-on

MP7212   Acknowledge with key: **0**
            Acknowledge automatically: **1**

ISO programming: Block number increment

MP7220   **0 to 150**

Length of file names

<div align="right">426</div>

MP7222   Maximum 8 characters: **0**
            Maximum 12 characters: **1**
            Maximum 16 characters: **2**

Inhibit particular file types

👆 | If a particular file type is inhibited, all files of this type will be erased.

MP7224.0 Do not inhibit file types: **+0**
             Inhibit HEIDENHAIN programs: **+1**
             Inhibit ISO programs: **+2**
             Inhibit tool tables: **+4**
             Inhibit datum tables: **+8**
             Inhibit pallet tables: **+16**
             Inhibit text files: **+32**
             Inhibit PNT tables: **+128**

Inhibit the editor for certain file types

MP7224.1 Do not inhibit editor: **+0**
             Inhibit editor for HEIDENHAIN programs: **+1**
             Inhibit editor for ISO programs: **+2**
             Inhibit editor for tool tables: **+4**
             Inhibit editor for datum tables: **+8**
             Inhibit editor for pallet tables: **+16**
             Inhibit editor for text files: **+32**
             Inhibit editor for PNT tables: **+128**

TNC displays, TNC editor

Configure pallet files

MP7226.0   Pallet file inactive: **0**
                Number of pallets per pallet file: **1 to 255**

Configure datum files

MP7226.1   Datum table inactive: **0**
                Number of datums per datum table: **1 to 255**

Dialog language for TNC 425/TNC 415 B/TNC 407

MP7230   German: **0**
                English: **1**

Dialog language for TNC 426

MP7230   English: **0**
                German: **1**
                Czech: **2**
                French: **3**
                Italian: **4**
                Spanish: **5**
                Portuguese: **6**
                Swedish: **7**
                Danish: **8**
                Finnish: **9**
                Dutch: **10**

Protect OEM cycles in the TNC program memory

426

MP7240   Protect programs: **0**
                Do not protect programs: **1**

Configure tool tables

MP7260   Inactive: **0**
                Number of tools per tool table: **1 to 254**

Configure pocket tables

MP7261   Inactive: **0**
                Number of pockets per pocket table: **1 to 254**

TNC displays, TNC editor

**Configure tool table; column number of the data in the tool table for
(Do not show data in the table: enter 0)**

MP7266.0	Tool name – NAME:	*0 to 24*
MP7266.1	Tool length – L:	*0 to 24*
MP7266.2	Tool radius – R:	*0 to 24*
MP7266.3	Tool radius – R2:	*0 to 24*
MP7266.4	Oversize length – DL:	*0 to 24*
MP7266.5	Oversize radius – DR:	*0 to 24*
MP7266.6	Oversize radius2 – DR2:	*0 to 24*
MP7266.7	Tool locked – TL:	*0 to 24*
MP7266.8	Replacement tool – RT:	*0 to 24*
MP7266.9	Maximum tool life – TIME1:	*0 to 24*
MP7266.10	Maximum tool life for TOOL CALL – TIME2:	*0 to 24*
MP7266.11	Current tool age – CUR. TIME:	*0 to 24*
MP7266.12	Tool comment – DOC:	*0 to 24*
MP7266.13	Number of cutting edges – CUT.:	*0 to 24*
MP7266.14	Length tolerance for tool wear – LTOL:	*0 to 24*
MP7266.15	Radius tolerance for tool wear – RTOL:	*0 to 24*
MP7266.16	Cutting direction – DIRECT.:	*0 to 24*
MP7266.17	PLC status – PLC:	*0 to 24*
MP7266.18	Tool offset in addition to MP6530 between upper edge of stylus and lower edge of tool – TT:L-OFFS.:	*0 to 24*
MP7266.19	Tool offset between stylus center and tool center – TT:R-OFFS.:	*0 to 24*
MP7266.20	Length tolerance for tool break – LBREAK:	*0 to 24*
MP7266.21	Radius tolerance for tool break – RBREAK:	*0 to 24*

Additional machine parameters for TNC 426

MP7266.22	Tooth length (Cycle 22) – LCUTS:	*0 to 24*
MP7266.23	Maximum plunge angle (Cycle 22) – ANGLE:	*0 to 24*

TNC displays, TNC editor

**Configure pocket table; column number of the data in the tool table for
(Do not show data in the table: enter 0)**

MP7267.0	Tool number – T:	**0 to 5**
MP7267.1	Special tool – ST:	**0 to 5**
MP7267.2	Fixed pocket – F:	**0 to 5**
MP7267.3	Pocket locked – L:	**0 to 5**
MP7267.4	PLC – Status – PLC:	**0 to 5**

MANUAL OPERATION mode: Display feed rate

MP7270   Display feed rate F only if an axis direction button is pressed: **0**
Display feed rate F even if no axis direction button is pressed
(feed rate of the "slowest" axis): **1**

Decimal character

MP7280   The decimal character is a comma: **0**
The decimal character is a point: **1**

Position display in the tool axis

MP7285   Display is referenced to the tool datum: **0**
Display in the tool axis is referenced to the tool face: **1**

Display step for the X axis

MP7290.0   0.1 mm: **0**
0.05 mm: **1**
0.01 mm: **2**
0.005 mm: **3**
0.001 mm: **4**
0.0005 mm: **5**
0.0001 mm: **6**

Display step for the Y axis

MP7290.1   0.1 mm: **0**
0.05 mm: **1**
0.01 mm: **2**
0.005 mm: **3**
0.001 mm: **4**
0.0005 mm: **5**
0.0001 mm: **6**

Display step for the Z axis

MP7290.2   0.1 mm: **0**
0.05 mm: **1**
0.01 mm: **2**
0.005 mm: **3**
0.001 mm: **4**
0.0005 mm: **5**
0.0001 mm: **6**

TNC displays, TNC editor

Display step for the IV axis

MP7290.3   0.1 mm: **0**
0.05 mm: **1**
0.01 mm: **2**
0.005 mm: **3**
0.001 mm: **4**
0.0005 mm: **5**
0.0001 mm: **6**

Display step for the V axis

MP7290.4   0.1 mm: **0**
0.05 mm: **1**
0.01 mm: **2**
0.005 mm: **3**
0.001 mm: **4**
0.0005 mm: **5**
0.0001 mm: **6**

Inhibit datum setting

MP7295   Do not inhibit datum setting: **+0**
Inhibit datum setting in the X axis: **+1**
Inhibit datum setting in the Y axis: **+2**
Inhibit datum setting in the Z axis: **+4**
Inhibit datum setting in the IV axis: **+8**
Inhibit datum setting in the V axis: **+16**

Inhibit datum setting with the orange axis keys

MP7296   Do not inhibit datum setting: **0**
Inhibit datum setting with the orange axis keys: **1**

Reset status display, Q parameters and tool data

MP7300   Reset them all when a program is selected: **0**
Reset them all when a program is selected and with M02, M30, N99999: **1**
Reset only status display and tool data when a program is selected: **2**
Reset only status display and tool data when a program is selected
and with M02, M30, N99999: **3**
Reset status display and Q parameters when a program is selected: **4**
Reset status display and Q parameters when a program is selected
and with M02, M30, N99999: **5**
Reset status display when a program is selected: **6**
Reset status display when a program is selected
and with M02, M30, N99999: **7**

Graphic display mode

MP7310   Projection in 3 planes according to ISO 6433, projection method 1: **+0**
Projection in 3 planes according to ISO 6433, projection method 2: **+1**

Do not rotate coordinate system for graphic display: **+0**
Rotate coordinate system for graphic display by 90°: **+2**

Display new BLK FORM in G54 DATUM SHIFT referenced to previous datum: **+0**
Display new BLK FORM in G54 DATUM SHIFT referenced to new datum: **+4**

Do not show cursor position during projection in 3 planes mode: **+0**
Show cursor position during projection in 3 planes mode: **+8**

TNC displays, TNC editor

Graphic simulation without programmed tool axis: Tool radius

MP7315   *0 to 99 999.9999 [mm]*

Graphic simulation without programmed tool axis: Penetration depth

MP7316   *0 to 99 999.9999 [mm]*

Graphic simulation without programmed tool axis: M function for start

MP7317.0   *0 to 88* (0: Function inactive)

Graphic simulation without programmed tool axis: M function for end

MP7317.1   *0 to 88* (0: Function inactive)

Determine the TNC program memory to be occupied during blockwise transfer   `426`

MP7228.0   Minimum program memory to be used: *1 to 1024 [kilobytes]*

MP7228.1   Maximum program memory to be used: *1 to 1024 [kilobytes]*

Screen saver   `407` `415` `425`

               Enter the time after which the TNC should start the screen saver

MP7392   *0 to 99 [min]* (0: Function inactive)

Machining and program run

Cycle G85 RIGID TAPPING: Oriented spindle stop at beginning of cycle

MP7160   Spindle orientation: **0**
No spindle orientation: **1**

Effect of Cycle G72 SCALING FACTOR

MP7410   SCALING FACTOR effective in 3 axes: **0**
SCALING FACTOR effective in the working plane only: **1**

Tool data in the programmable probe Cycle G55

MP7411   Overwrite current tool data by the calibrated data from the 3D touch probe system: **0**
Current tool data are retained: **1**

Cycle G57 ROUGH-OUT: Selecting the pocket milling technique

MP7420   Mill a channel around the contour—clockwise for islands and counterclockwise for pockets: **+0**

Mill a channel around the contour—clockwise for pockets and counterclockwise for islands: **+1**

First mill the channel, then rough out the pocket: **+0**
First rough out the pocket, then mill the channel: **+2**

Combine compensated contours: **+0**
Combine uncompensated contours: **+4**

Complete one process for all infeeds before switching to the other process: **+0**
Mill channel and rough-out for each infeed depth before continuing to the next depth: **+8**

POCKET MILLING cycle (G75/G76) and CIRCULAR POCKET cycle (G77/G78): Overlap factor

MP7430   **0.1 to 1.414**

Permissible distance by which an end point can be off the path of a perfect circle

MP7431   **0.0001 to 0.016 [mm]**

Behavior of M functions

MP7440   Program stop with M06: **+0**
No program stop with M06: **+1**

No cycle call with M89: **+0**
Modal cycle call with M89: **+2**

Program stop with M functions: **+0**
No program stop with M functions: **+4**

Kv factors cannot be switched through M105 and M106: **+0**
Kv factors can be switched through M105 and M106: **+8**

Reduce the feed rate in the tool axis with M103 F.. , function inactive: **+0**
Reduce the feed rate in the tool axis with M103 F.. , function active: **+16**

> The Kv factors for position loop gain are set by the machine tool builder. For more detailed information on this subject, refer to your machine manual.

Machining and program run

Corners whose angles are less than the entered value will be machined at a decelerated feed rate (if radius compensation is R0 or if the angle is at an inside corner)

> This feature works both during operation with servo lag as well as with feed precontrol.

MP7460   *0.0000 to 179.9999 [°]*

Maximum contouring speed at a feed rate override setting of 100% in the program run modes   `407` `415` `425`

MP7470   *0 to 99 999 [mm/min]*

Datums from a datum table are referenced to the

MP7475   Workpiece datum: *0*
             Machine datum: *1*

Electronic handwheel

Handwheel type

MP7640   Machine without handwheel: **0**
HR 330 with additional keys—the handwheel keys for traverse direction and
rapid traverse are evaluated by the NC: **1**
HR 130 without additional keys: **2**
HR 330 with additional keys—the handwheel keys for traverse direction and
rapid traverse are evaluated by the PLC: **3**
HR 332 with twelve additional keys: **4**
Multi-axis handwheel with additional keys: **5**
HR 410 with auxiliary functions: **6**

Entry of interpolation factor

MP7641   Interpolation factor is entered on the keyboard: **0**
Interpolation factor is set by the PLC: **1**

Machine parameters that can be set for the handwheel by the machine tool builder

MP 7645.0	*0 to 255*
MP 7645.1	*0 to 255*
MP 7645.2	*0 to 255*
MP 7645.3	*0 to 255*
MP 7645.4	*0 to 255*
MP 7645.5	*0 to 255*
MP 7645.6	*0 to 255*
MP 7645.7	*0 to 255*

11.2 Miscellaneous Functions (M Functions)

Miscellaneous functions with predetermined effect

M	Effect	Effective at block start	end	Page
M00	Stop program run/spindle STOP/coolant OFF		•	3-5
M02	Stop program run/spindle STOP/coolant OFF/ Clear status display (depending on machine parameter)/ Go to block 1		•	3-5
M03	Spindle ON clockwise	•		
M04	Spindle ON counterclockwise	•		
M05	Spindle STOP		•	
M06	Tool change/stop program run (depending on machine parameter)/ Spindle STOP		•	3-5
M08	Coolant ON	•		
M09	Coolant OFF		•	
M13	Spindle ON clockwise/coolant ON	•		
M14	Spindle ON counterclockwise/coolant ON	•		
M30	Same function as M02		•	3-5
M89	Vacant miscellaneous function **or** Cycle call, modally effective (depending on machine parameter)	•	•	8-3
M90	Only in lag mode: Constant contouring speed at corners	•		5-36
M91	Within the positioning block: Coordinates are referenced to machine datum	•		5-39
M92	Within the positioning block: Coordinates are referenced to position defined by machine builder, such as tool change position	•		5-39
M93	Reserved	•		
M94	Reduce display of rotary axis to value less than 360°	•		5-43
M95	Reserved		•	
M96	Reserved		•	
M97	Machine small contour steps		•	5-37
M98	Completely machine open contours		•	5-38
M99	Blockwise cycle call		•	8-3

Miscellaneous functions with predetermined effect

M	Effect	Effective at block start	end	Page
M101	Automatic tool change with replacement tool if maximum tool life has expired	•		4-16
M102	Reset M101		•	4-16
M103	Reduce feed rate during plunging to factor F (percentage)	•		5-40
M105	Machining with first Kv factor	•		
M106	Machining with second Kv factor	•		
M107	Suppress error message for replacement tool with oversize	•		
M108	Reset M107		•	
M109	Constant contouring speed at tool cutting edge on circular arcs (increase and decrease feed rate)	•		5-41
M110	Constant contouring speed at tool cutting edge on circular arcs (feed rate decrease only)	•		5-41
M111	Reset M109/M110		•	5-41
M112	Automatic insertion of rounding arcs at non-tangential straight-line transitions; Enter tolerance T for contour deviation	•		5-41
M113	Reset M112		•	
M114	Automatic compensation of machine geometry during operation with tilting axes	•		5-42
M115	Reset M114		•	
M116	Feed rate for angular axes in mm/min	•		5-43
M126	Optimized traverse of rotary axes	•		5-44
M127	Reset M126		•	5-44

> The miscellaneous functions M105 and M106 are defined and enabled by the machine tool builder.
> Please refer to your machine manual for more information.

Vacant miscellaneous functions

Vacant miscellaneous functions are set by the machine tool builder.
These functions are described in the machine manual.

M	Function	Effective at Start of block	Effective at End	M	Function	Effective at Start of block	Effective at End
M01			•	M50		•	
M07		•		M51		•	
M10			•	M52			•
M11		•		M53			•
M12			•	M54			•
M15		•		M55		•	
M16		•		M56		•	
M17		•		M57		•	
M18		•		M58		•	
M19			•	M59		•	
M20		•		M60			•
M21		•		M61		•	
M22		•		M62		•	
M23		•		M63			•
M24		•		M64			•
M25		•		M65			•
M26		•		M66			•
M27		•		M67			•
M28		•		M68			•
M29		•		M69			•
M31		•		M70			•
M32			•	M71		•	
M33			•	M72		•	
M34			•	M73		•	
M35			•	M74		•	
M36		•		M75		•	
M37		•		M76		•	
M38		•		M77		•	
M39		•		M78		•	
M40		•		M79		•	
M41		•		M80		•	
M42		•		M81		•	
M43		•		M82		•	
M44		•		M83		•	
M45		•		M84		•	
M46		•		M85		•	
M47		•		M86		•	
M48		•		M87		•	
M49		•		M88		•	

11.3 Preassigned Q Parameters

Q100 to Q113 are assigned values by the TNC. These values include:

- Values from the PLC
- Tool and spindle data
- Data on operating status, etc.

Values from the PLC: Q100 to Q107

The TNC uses Q100 to Q107 to transfer values from the PLC to an NC program.

Tool radius: Q108

The current value of the tool radius is assigned to Q108.

Tool axis: Q109

The value of Q109 depends on the current tool axis.

Tool axis	Parameter value
No tool axis defined	Q109 = −1
Z axis	Q109 = 2
Y axis	Q109 = 1
X axis	Q109 = 0

Spindle status: Q110

The value of Q110 depends on which M function was last programmed.

M function	Parameter value
No spindle status defined	Q110 = −1
M03: Spindle ON, clockwise	Q110 = 0
M04: Spindle ON, counterclockwise	Q110 = 1
M05 after M03	Q110 = 2
M05 after M04	Q110 = 3

Coolant on/off: Q111

M function	Parameter value
M08: Coolant on	Q111 = 1
M09: Coolant off	Q111 = 0

Overlap factor: Q112

The overlap factor for pocket milling (MP 7430) is assigned to Q112.

Unit of measurement for dimensions in the part program: Q113

The value of parameter Q113 specifies whether the highest-level NC program (for nesting with %) is programmed in millimeters or inches.

Dimensions of the main program	Parameter value
Metric system (mm)	Q113 = 0
Inch system	Q113 = 1

Tool length: Q114

The current value for the tool length is assigned to Q114.

Coordinates after probing during program run

Q115 to Q119 contain the coordinates of the spindle position at the moment of contact during programmed measurement with the 3D touch probe. The length and radius of the probe tip are not compensated in these coordinates.

Coordinate axis	Parameter
X axis	Q115
Y axis	Q116
Z axis	Q117
IVth axis	Q118
Vth axis	Q119

Deviation between actual value and nominal value during automatic tool measurement with the TT 110 touch probe (*conversational programming only*)

Actual-nominal deviation	Parameter
Tool length	Q115
Tool radius	Q116

Tilting the working plane with mathematical angles: Rotary axis coordinates calculated by the TNC

Coordinates	Parameter
A axis	Q120
B axis	Q121
C axis	Q122

407 **415** **425**

11.4 Features, Specifications and Accessories

Technical information	
Description	Contouring control for machines with up to five axes. Features digital speed control and oriented spindle stop.
Components	Logic unit, keyboard, color VDU with soft keys
Data interfaces	RS-232-C / V.24 RS-422 / V.11
	Expanded data interface with LSV/2 protocol for remote operation of the TNC through the data interface with HEIDEN-HAIN software TNC REMOTE.
Simultaneous axis control for contour elements	Straight lines: up to 5 axes TNC 407: 3 axes Export versions TNC 415 F, TNC 425 E and TNC 426 E: 4 axes
	Circles: up to 3 axes (with tilted working plane)
	Helices: 3 axes
"Look Ahead"	Defined rounding of discontinuous contour transitions (such as for 3D surfaces) Collision prevention with the SL cycle for open contours (G125) Geometry pre-calculation for feed rate adaptation
Background programming	One part program can be edited while the TNC runs another program (TNC 407: without graphics).
Graphics	Interactive programming graphics Test run graphics Simultaneous program run graphics (not with TNC 407)
File types	HEIDENHAIN conversational programming ISO programming Tool tables Datum tables Point tables Pallet files Text files System files
Program memory	**TNC 426:** 170 MB hard disk for NC programs No limit on number of files **TNC 425, TNC 415 B:** Battery-buffered, for up to 100 files Capacity: 256K bytes **TNC 407:** Capacity: 128K bytes
Tool definitions	Up to 254 tools in the program or in tables
Programming support	Functions for approaching and departing the contour Structuring long programs **Additional feature on TNC 426:** On-screen pocket calculator

Programmable functions	
Contour elements	Straight line Chamfer Circular arc Circle center Circle radius Tangentially connecting circular arc Corner rounding Straight lines and circular arcs for approaching and departing contours
Free contour programming	For all contour elements not dimensioned for conventional NC programming
Three-dimensional radius compensation (not on TNC 407)	For changing tool data without having to recalculate the part program
Program jumps	Subprograms Program section repeats Main program as subprogram
Fixed cycles	Peck drilling and tapping (also with synchronized spindle) Thread cutting Rectangular and circular pocket milling Slot milling Milling pockets and islands from a list of subcontour elements Cylindrical surface interpolation
Coordinate transformations	Datum shift Mirroring Rotation Scaling factor Tilting the working plane (not TNC 407)
3D touch probe applications	Touch probe functions for setting datums and for automatic workpiece measurement Digitizing 3D surfaces with the measuring touch probe (optional, not TNC 407) Digitizing 3D surfaces with the triggering touch probe (optional, conversational programming only) Automatic tool measurement with the TT 120 touch probe (conversational programming only)
Mathematical functions	Basic arithmetic ($+$, $-$, x, $\div$) Trigonometry (sine, cosine, tangent, arc sine, arc cosine, arc tangent) Square root ($\sqrt{a}$) and Root sum of squares ($\sqrt{a^2 + b^2}$) Squaring (SQ) Powers (^) π Logarithm Exponential function Negation (NEG) Forming an absolute number (ABS) Forming an integer (INT) Dropping the values before the decimal point (FRAC) Logical comparisons (greater than, less than, equal to, not equal to)

11.4 Features, Specifications and Accessories

TNC Specifications		
Block processing time	4 milliseconds per block (TNC 407: 25 ms/block)	
Control loop cycle time	TNC 426:	Contouring interpolation: 3 ms Fine interpolation: 0.6 ms (speed)
	TNC 425:	Contouring interpolation: 3 ms Fine interpolation: 0.6 ms (speed)
	TNC 415 B:	Contouring interpolation: 3 ms Fine interpolation: 0.6 ms (contour)
	TNC 407:	6 ms
Data transfer rate	TNC 426:	Max. 115 200 baud
	TNC 425:	Max. 38 400 baud
	TNC 415 B:	Max. 38 400 baud
	TNC 407:	Max. 38 400 baud
Ambient temperature	Operation: 0° to 45° C (32° to 113° F) Storage: –30° to 70° C (–22° to 158° F)	
Traverse range	Max. ± 100 m (± 2540 in.)	
Traversing speed	Max. 300 m/min (11 810 ipm)	
Spindle speed	Max. 99 999 rpm	
Input range	Min. 0.1 µm (0.00001 in.) or 0.0001° (TNC 407, TNC 415 F, TNC 425 E, TNC 426 E: 1 µm) Max. 99 999.999 mm (3937 in.) or 99 999.999°	

Accessories

FE 401 floppy disk unit	
Description	Portable table-top unit
Applications	All TNC contouring controls as well as TNC 131, TNC 135
Data interfaces	2 interfaces RS-232-C/V.24
Data transfer rate	• TNC: 2400 to 38 400 baud • PRT: 110 to 9600 baud
Disk drives	Separate drive for copying, capacity 795 kilobytes (approx. 25,000 blocks), up to 256 files
Floppy disks	3.5 inch, DS DD, 135 TPI

TS 220 and TS 630 triggering 3D touch probes		
Description		Touch probe system with ruby tip and stylus with rated break point, standard shank for spindle insertion
Signal transmission	TS 220:	Transmission via cable, integrated interface
	TS 630:	Infrared transmission, separate transmitting and receiving units
Spindle insertion	TS 220:	manual
	TS 630:	automatic
Probing reproducibility		Better than 1 µm (0.000 04 in.)
Probing speed		Maximum 3 m/min (118 ipm)

TT 120 triggering 3D touch probe	
Description	Touch probe system with hardened, stainless-steel probing element (steel plate), protection class IP 67
Interface	Connected to TNC via 5 V supply
Installation	Fixed installation within the machine working space
Probing speed	Maximum 3m/min (118 ipm)

11.4 Features, Specifications and Accessories

Electronic handwheels	
HR 130	Integrable unit
HR 150	Fixed-axis handwheel for the HRA 110 adapter
HR 330	Portable version, transmission via cable. Includes axis address keys, rapid traverse key, safety switch, emergency stop button

11.5 TNC Error Messages

The TNC automatically generates error messages when it detects problems such as

- Incorrect data input
- Logical errors in the program
- Contour elements that are impossible to machine
- Incorrect use of the touch probe system

An error message that contains a program block number was caused by an error in the indicated block or in the preceding block. To clear a TNC error message, first correct the error and then press the CE key.

Some of the more frequent TNC error messages are explained in the following list.

TNC error messages during programming

> **ENTRY VALUE INCORRECT**

- Enter a correct label number
- Note the input limits

> **EXT. IN-/OUTPUT NOT READY**

- Connect the data transfer cable.
- Transfer cable is defective or not soldered properly.
- Switch on the connected device (PC, printer).
- The data transfer speeds (baud rates) are not identical.

> **FILE FORMAT HAS CHANGED**

After a software change the internal format has changed. The TNC can no longer read the file. Erase the file.

407   **415**   **425**

> **FURTHER PROGRAM ENTRY IMPOSSIBLE**

Erase some old files to make room for new ones.

> **JUMP TO LABEL 0 NOT PERMITTED**

Do not program L 0,0.

> **LABEL NUMBER ALREADY ASSIGNED**

A given label number can only be entered once in a program.

> **PROTECTED PGM !**

Cancel edit protection if you wish to edit the program.

TNC error messages during test run and program run

ANGLE REFERENCE MISSING

• Complete your definition of the arc and its end points.
• If you enter polar coordinates, define the polar coordinate angle correctly.

ARITHMETICAL ERROR

You have calculated with non-permissible values.

• Define values within the range limits
• Choose probe positions for the 3D touch probe that are farther apart
• All calculations must be mathematically possible

AXIS DOUBLE PROGRAMMED

Each axis can have only one value for position coordinates.

BLK FORM DEFINITION INCORRECT

• Program the MIN and MAX points according to the instructions.
• Choose a ratio of sides that is less than 200:1.

CHAMFER NOT PERMITTED

A chamfer block must be located between two straight line blocks with identical radius compensation.

CIRCLE CENTER UNDEFINED

• Define a circle center with I,J (JK, IK).
• Define a pole with I,J (JK, IK).

CIRCLE END POS. INCORRECT

• Enter complete information for connecting arc.
• Enter end points that lie on the circular path.

CYCL INCOMPLETE

• Define the cycles with all data in the proper sequence.
• Do not call the coordinate transformation cycles.
• Define a cycle before calling it.
• Enter a pecking depth other than 0.

11.5 TNC Error Messages

EXCESSIVE SUBPROGRAMMING

- Conclude subprograms with G98 L0.
- Program Ln,0 for subprogram calls.
- Program Ln,m for program section repeats.
- Subprograms cannot call themselves.
- Subprograms cannot be nested in more than eight levels.
- Main programs cannot be nested as subprograms in more than four levels.

FEED RATE IS MISSING

- Enter feed rate for G01 block.

GROSS POSITIONING ERROR

The TNC monitors positions and movements. If the actual position deviates excessively from the nominal position, this blinking error message is displayed. You must press the END key for a few seconds to correct the error.

KEY NON-FUNCTIONAL

This message always appears when you press a key that is not needed for the current dialog.

LABEL NUMBER NOT FOUND

Call only label numbers that have been set.

NO EDITING OF RUNNING PROGRAM

A program cannot be edited while it is being transmitted or executed.

PATH OFFSET WRONGLY ENDED

Do not cancel tool radius compensation in a block with a circular path.

PATH OFFSET WRONGLY STARTED

- Use the same radius compensation before and after a G24 and G25 block.
- Do not begin tool radius compensation in a block with a circular path.

PGM-SECTION CANNOT BE SHOWN

- Enter a smaller tool radius.
- 4D and 5D movements cannot be graphically simulated.
- Enter a tool axis for simulation that is the same as the axis in the definition of the workpiece blank.

PLANE WRONGLY DEFINED

- Do not change the tool axis while a basic rotation is active.
- Correctly define the main axes for circular arcs.
- Define both main axes for I,J (JK, IK).

PROBE SYSTEM NOT READY

- Be sure the transmitting/receiving window of the TS 630 is oriented to the receiving unit.
- Check whether the touch probe is ready for operation.

PROGRAM-START UNDEFINED

- Begin the program only with a G99 block.
- Do not resume an interrupted program at a block with a tangential arc or if a previously defined pole is needed.
- Program the first block with axis motion with G00 G40 G90.

RADIUS COMP. UNDEFINED

Enter radius compensation G41 or G42 in the first subprogram for Cycle G37 CONTOUR GEOMETRY.

ROUNDING-OFF NOT PERMITTED

Enter tangentially connecting arcs and rounding arcs correctly.

ROUNDING RADIUS TOO LARGE

Rounding arcs must fit between contour elements.

11.5 TNC Error Messages

SELECTED BLOCK NOT ADDRESSED

Before a test run or program run, you must enter GOTO 0.

STYLUS ALREADY IN CONTACT

Before probing, pre-position the stylus where it is not touching the
workpiece surface.

TOOL RADIUS TOO LARGE

Enter a tool radius that
• lies within the given limits
• permits the contour elements to be calculated and machined.

TOUCH POINT INACCESSIBLE

Pre-position the 3D touch probe to a position nearer the model.

WRONG AXIS PROGRAMMED

• Do not attempt to program locked axes.
• Program a rectangular pocket or slot in the working plane.
• Do not mirror rotary axes.
• Enter a positive chamfer length.

WRONG RPM

Program a spindle speed within the permissible range.

WRONG SIGN PROGRAMMED

Enter the correct sign for the cycle parameter.

11.6 Address Letters (ISO)

G Functions

Group	G	Function	Non-modal function	See Page
Positioning	00	Straight line interpolation, Cartesian coordinates, rapid traverse		5-10
	01	Straight line interpolation, Cartesian coordinates		5-10
	02	Circular interpolation, Cartesian coordinates, clockwise		5-18
	03	Circular interpolation, Cartesian coordinates, counterclockwise		5-18
	05	Circular interpolation, Cartesian, no direction of rotation given		5-18
	06	Circular interpolation, Cartesian, tangential contour transition		5-24
	07	Paraxial positioning block	•	
	10	Straight line interpolation, polar coordinates, rapid traverse		5-28
	11	Straight line interpolation, polar coordinates		5-28
	12	Circular interpolation, polar coordinates, clockwise		5-30
	13	Circular interpolation, polar coordinates, counterclockwise		5-30
	15	Circular interpolation, polar, no direction of rotation given		5-30
	16	Circular interpolation, polar, tangential contour transition		5-32
Cycles	04	Dwell time	•	8-53
	28	Mirror image		8-48
	36	Oriented spindle stop		8-54
	37	Definition of the contour geometry		8-18
	39	Program call, cycle call with G79	•	8-53
	53	Datum shift in datum table		8-45
	54	Datum shift in program		8-43
	56	Pilot drilling (in connection with G37) SLI		8-27
	57	Rough-out (in connection with G37) SLI		8-19
	58	Contour milling, clockwise (in connection with G37) SLI		8-28
	59	Contour milling, counterclockwise (in connection with G37) SLI		8-28
	72	Scaling factor		8-51
	73	Rotation of the coordinate system		8-50
	74	Slot milling		8-11
	75	Rectangular pocket milling, clockwise		8-13
	76	Rectangular pocket milling, counterclockwise		8-13
	77	Circular pocket milling, clockwise		8-15
	78	Circular pocket milling, counterclockwise		8-15
	80	Working plane		8-55
	83	Pecking		8-4
	84	Tapping with floating tap holder		8-6
	85	Rigid tapping		8-8
	86	Thread cutting		8-9
	120	Contour data		8-32
	121	Pilot drilling (in connection with G37) SLII		8-33
	122	Rough-out (in connection with G37) SLII		8-34
	123	Floor finishing (in connection with G37) SLII		8-34
	124	Side finishing (in connection with G37) SLII		8-35
	125	Contour train (in connection with G37) SLII		8-37
	127	Cylinder surface (in connection with G37) SLII		8-39
	79	Cycle call	•	8-3
Select working plane	17	Working plane XY, tool axis Z		5-16
	18	Working plane ZX, tool axis Y		5-16
	19	Working plane YZ, tool axis X		5-16
	20	Tool axis: IV		5-16
	24	Chamfer with length R		5-13
	25	Corner rounding with R		5-26
	26	Tangential contour approach with R		5-6
	27	Tangential contour departure with R		5-6
	29	Transfer the last nominal position value as pole		5-16

Group	G	Function	Non-modal function	See Page
Define blank form	30	Blank form definition for graphics, MIN point		4-21
	31	Blank form definition for graphics, MAX point		4-21
	38	Stop program run		3-5
Tool path compensation	40	No tool radius compensation		4-18
	41	Tool radius compensation, left of the contour (RL)		4-18
	42	Tool radius compensation, right of the contour (RR)		4-18
	43	Paraxial compensation, lengthening (R+)		4-18
	44	Paraxial compensation, shortening (R–)		4-18
	51	Next tool number (with central tool file)	•	4-15
	55	Programmable probing function		7-23
Unit of measurement	70	Inches (at start of program)		4-21
	71	Millimeters (at start of program)		4-21
Dimensioning	90	Absolute dimensions		1-15
	91	Incremental dimensions		1-15
	98	Set label number	•	6-2
	99	Tool definition in program	•	4-7

Address letter	Function
%	Beginning of program or program call with G39
A	Rotary motion about the X axis
B	Rotary motion about the Y axis
C	Rotary motion about the Z axis
D	Parameter definition (program parameter Q)
D	OEM cycle definition (not on TNC 426)
E	Tolerance for rounding arc with M112
F	Feed rate
F	Dwell time with G04
F	Scaling factor with G72
G	Preparatory function
H	Angle for polar coordinates in incremental/absolute dimensions
H	Rotational angle with G73
I	X coordinate of circle center/pole
J	Y coordinate of circle center/pole
K	Z coordinate of circle center/pole
L	Set label number with G98
L	Go to label number
L	Tool length with G99
M	Miscellaneous function
N	Block number
P	Cycle parameter in fixed cycles
P	Parameter in parameter definitions
Q	Program parameter/cycle parameter Q
R	Polar coordinate radius
R	Circle radius with G02/G03/G05
R	Rounding radius with G25/G26/G27
R	Chamfer with G24
R	Tool radius with G99
S	Spindle speed
S	Oriented spindle stop with G36
T	Tool definition with G99
T	Tool call
U	Linear motion parallel to the X axis
V	Linear motion parallel to the Y axis
W	Linear motion parallel to the Z axis
X	X axis
Y	Y axis
Z	Z axis
*	End of block

11.6 Address Letters (ISO)

Parameter definitions

D	Function	See page
00	Assignment	7-7
01	Addition	7-7
02	Subtraction	7-7
03	Multiplication	7-7
04	Division	7-7
05	Square root	7-7
06	Sine	7-10
07	Cosine	7-10
08	Root sum of squares ($c = \sqrt{a^2 + b^2}$)	7-10
09	If equal, jump	7-11
10	If not equal, jump	7-11
11	If greater than, jump	7-11
12	If less than, jump	7-11
13	Angle (angle from $c \cdot \sin a$ and $c \cdot \cos a$)	7-10
14	Error number	7-14
15	Print	7-16
16	F-PRINT	7-17
18	Reading system data	7-18
19	Assignment PLC marker	7-19

Sequence of Program Steps
Milling an outside corner

Program step		Key/Function	Section in manual
1	**Open or select program**		4.4
	Entries: Program name	PGM MGT PGM NAME	
2	**Blank form for graphic displays**	G30/G31	4.4
3	**Define tools**	G99	
	Entries: Tool number	T...	
	Tool length	L...	
	Tool radius	R...	4.2
4	**Call tool data**	T...	
	Entries: Tool number		
	Spindle axis	G17	
	Spindle speed	S...	4.2
5	**Tool change**		
	Entries: Feed rate (rapid traverse)	G00	
	Radius compensation	G40	
	Absolute dimensions	G90	
	Coordinates of tool change position	X... Y... Z...	
	Miscellaneous function (tool change)	M06	e.g. 5.4
6	**Approach starting position**		
	Entries: Feed rate (rapid traverse)	G00	
	Radius compensation	G40	
	Coordinates of starting position	X... Y... Z...	
	Miscellaneous function (spindle ON clockwise)	M03	5.2/5.4
7	**Move tool axis to (first) working depth**		
	Entries: Feed rate (rapid traverse)	G00	
	Coordinate of the (first) working depth	Z...	5.4
8	**Approach contour**		
	Entries: Straight line interpolation	G01	
	Radius compensation for machining	G41/G42	
	Coordinates of first contour point	X... Y...	
	Machining feed rate	F...	
	For smooth approach (if necessary), program G26 after this block.		5.2/5.4
9	**Machining to last contour point**		
	Entries: Enter all required data for each contour element		5 to 8
10	**Depart contour**		
	Entries: Feed rate (rapid traverse)	G00	
	Cancel radius compensation	G40	
	Coordinates of end position	X... Y...	
	Miscellaneous function (spindle stop)	M05	
	For smooth departure (if necessary), program G27 after this block.		5.2/5.4
11	**Retract tool**		
	Entries: Feed rate (rapid traverse)	G00	
	Coordinate above workpiece surface	Z...	
	Miscellaneous function (end of program)	M02	5.2/5.4
12	**End of program**		

Miscellaneous Functions (M Functions)

Miscellaneous Functions with predetermined effect

M	Effect	Effective at block start	end	Page
M00	Program run/spindle STOP/coolant OFF		•	3-5
M02	Program run/spindle STOP/coolant OFF/ Clear status display (depending on machine parameter)/ Go to block 1		•	3-5
M03 M04 M05	Spindle ON clockwise Spindle ON counterclockwise Spindle STOP	• •	 •	
M06	Tool change/stop program run (depending on machine parameter)/ Spindle STOP		•	3-5
M08 M09	Coolant ON Coolant OFF	•	 •	
M13 M14	Spindle ON clockwise/coolant ON Spindle ON counterclockwise/coolant ON	• •		
M30	Same function as M02		•	3-5
M89	Vacant miscellaneous function **or** Cycle call, modally effective (depending on machine parameter)	•	 •	 8-3
M90	Only in lag mode: Constant contouring speed at corners	•		5-36
M91	Within the positioning block: Coordinates are referenced to machine datum	•		5-39
M92	Within the positioning block: Coordinates are referenced to position defined by machine builder, such as tool change position	•		5-39
M93	Reserved	•		
M94	Reduce display of rotary axis to value less than 360°	•		5-43
M95	Reserved		•	
M96	Reserved		•	
M97	Machine small contour steps		•	5-37
M98	Completely machine open contours		•	5-38
M99	Blockwise cycle call		•	8-3
M101 M102	Automatic tool change with replacement tool if maximum tool life has expired Reset M101	•	 •	4-16 4-16
M103	Reduce feed rate during plunging to factor F (percentage)	•		5-40
M105 M106	Machining with first Kv factor Machining with second Kv factor	• •		
M107 M108	Suppress error message for replacement tool with oversize Reset M107	•	 •	
M109 M110 M111	Constant contouring speed at tool cutting edge on circular arcs (increase and decrease feed rate) Constant contouring speed at tool cutting edge on circular arcs (feed rate decrease only) Reset M109/M110	• •	 •	5-41 5-41 5-41
M112 M113	Automatic insertion of rounding arcs at non-tangential straight-line transitions; Enter tolerance T for contour deviation Reset M112	•	 •	5-41
M114 M115	Automatic compensation of machine geometry during operation with tilting axes Reset M114	•	 •	5-42
M116	Feed rate for angular axes in mm/min	•		5-43
M126 M127	Optimized traverse of rotary axes Reset M126	•	 •	5-44

The miscellaneous functions M105 and M106 are defined and enabled by the machine tool builder.
Please refer to your machine manual for more information.