HEIDENHAIN

User's Manual
ISO Programming

TNC 425
TNC 415 B
TNC 407

June 1994

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	—	11.4
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 	5 to 8 or 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

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

Controls on the visual display unit

	Toggle display between machining and programming modes
GRAPHICS TEXT SPLIT SCREEN	Switch-over key for displaying graphics only, program blocks only, or both program blocks and graphics
	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

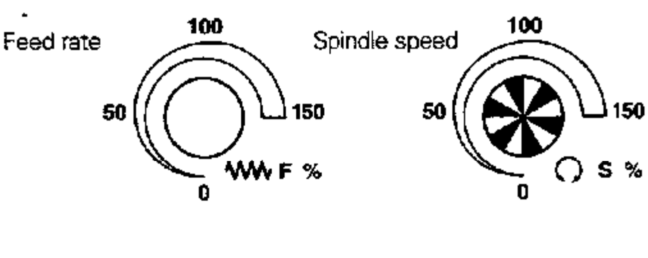
Program and file management

PGM NAME	Select programs and files
CL PGM	Delete programs and files
PGM CALL	Enter program call in a program *(conversational programming only)*
EXT	External data transfer
MOD	Miscellaneous functions

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



Programming path movements *(conversational programming only)*

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

Tool functions *(conversational programming only)*

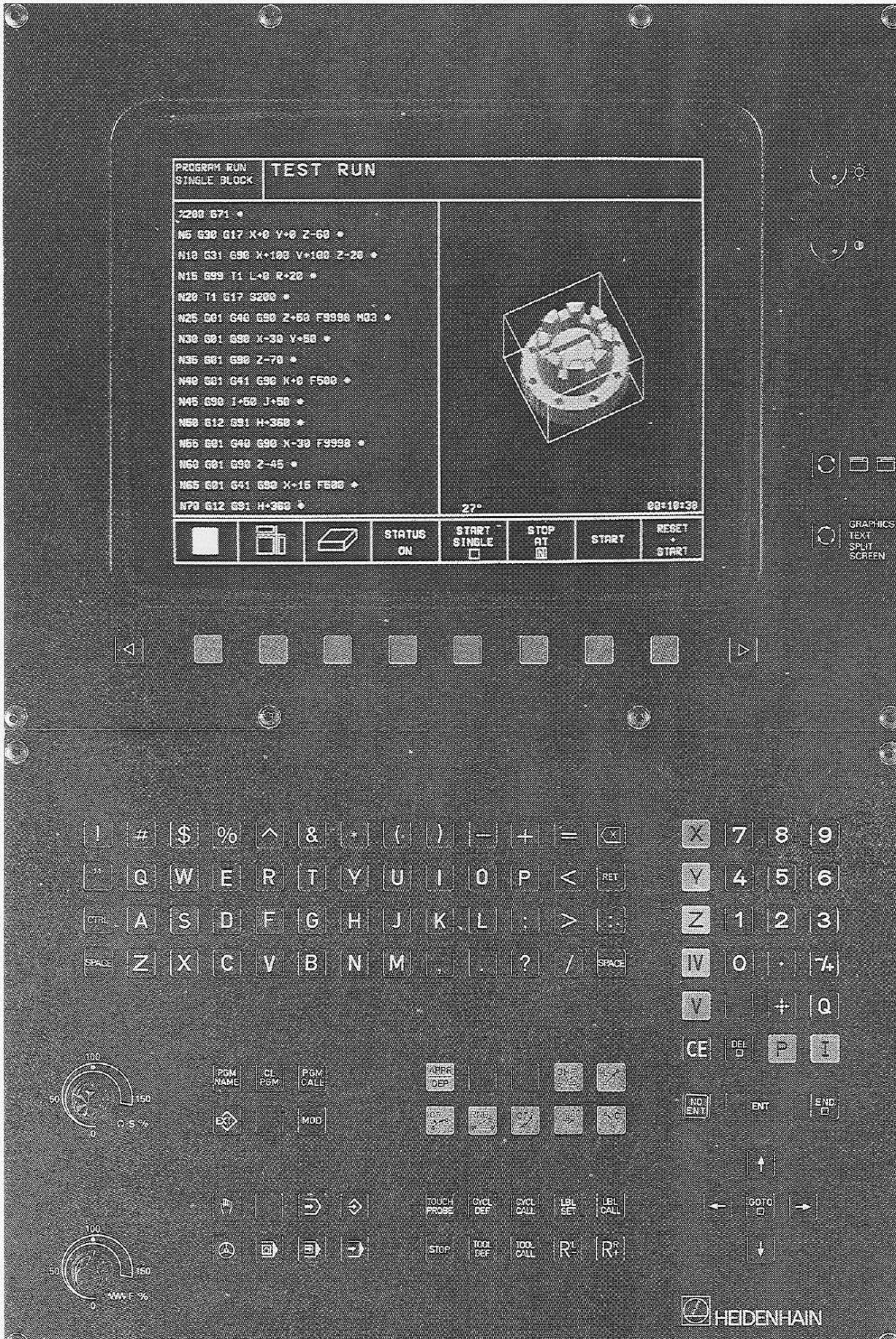
TOOL DEF / TOOL CALL	Enter or call tool length and radius
R^R_+ R^L_-	Activate tool radius compensation

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 *(conversational programming only)*
I	Incremental dimensions
Q	Q parameters for part families or mathematical functions *(conversational programming only)*
+	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


PROGRAM RUN
SINGLE BLOCK
TEST RUN
STATUS ON
START SINGLE
STOP AT
START
RESET + START
GRAPHICS
TEXT
SPLIT SCREEN
HEIDENHAIN

How to use this manual

This manual describes functions and features available on TNCs with the following NC software numbers or higher:

TNC model	NC software
TNC 407	243 030 10
TNC 415 B, TNC 425	259 930 10
TNC 415 F, TNC 425 E	259 940 10

The suffixes *E* and *F* identify export versions of the TNC.

The following functions are not available on the TNC 407:

- Graphics during program run
- Simultaneous linear movement in more than three axes

The export versions TNC 415 F and TNC 425 E have the following limitations:

- Input and machining accuracy are limited to 1 µm
- Simultaneous linear movement in no more than 3 axes

The versions otherwise differ only in technical details such as the type of speed control, block execution time, control loop cycle time and memory capacity.

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

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

- Probing functions for the 3D touch probe
- Rigid tapping
- Re-approaching a contour after an interruption

If you think a function may be unavailable because of a defect, please contact the machine tool builder.

This manual is intended for both TNC newcomers and experienced users.

If you're new to TNC, you can use the User's Manual 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 given can be practiced directly on the TNC. Each function is explained thoroughly when it is used for the first time.

As a beginner you should work through this manual completely from beginning to end to ensure that you are capable of fully exploiting the features of this powerful tool.

If you're already familiar with TNC, you can use the manual as a comprehensive reference and review guide. The table of contents and numerous cross-references help you quickly find the topics and information you need. Easy-to-read dialog flowcharts show you how to enter data for the desired function.

A description of the function of each key is provided in a box to the right of the key. 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 shown shaded in the flowcharts.

Dialog flowcharts

Dialog initiation

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 functions of the keys

NEXT DIALOG PROMPT	
+ Press this key	Function of the key
− Or press this key	Function of the alternative key

A dashed line indicates that you can press either the key(s) above the line or below it.

The trail of points means that:

- only part of the dialog is shown, or
- the dialog continues on the next page.

Contents User's Manual

TNC 407, TNC 415 B, TNC 425
(243 030-xx, 259 930-xx, 259 040-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 Introduction

1.1 The TNC 425, TNC 415 B and TNC 407

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 425

The TNC 425 features digital control of machine axis speed. This provides 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. The programming and machining functions of the TNC 425 are also provided on the TNC 407, with the following exceptions:

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

Technical differences between TNCs

	TNC 425	TNC 415 B	TNC 407
Speed control	Digital	Analog	Analog
Block execution time	4 ms	4 ms	24 ms
Control loop cycle time • Position controller	3 ms	2 ms	6 ms
Control loop cycle time • Speed controller	0.6 ms	—	—
Program memory	256K byte	256K byte	128K byte
Input resolution	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

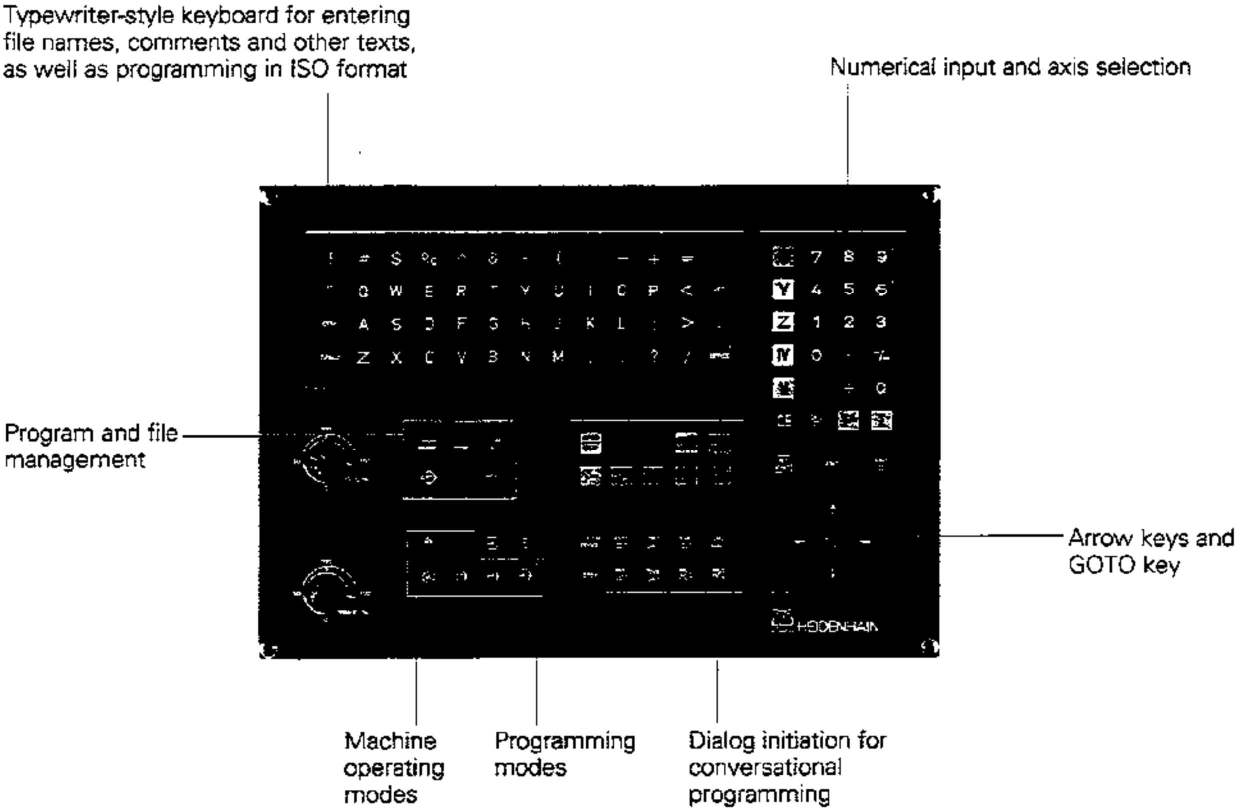
Workpiece machining can be graphically simulated both during machining (TNC 415 B and TNC 425 only) or before actual machining. Various display modes are available.

Compatibility

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

Keyboard

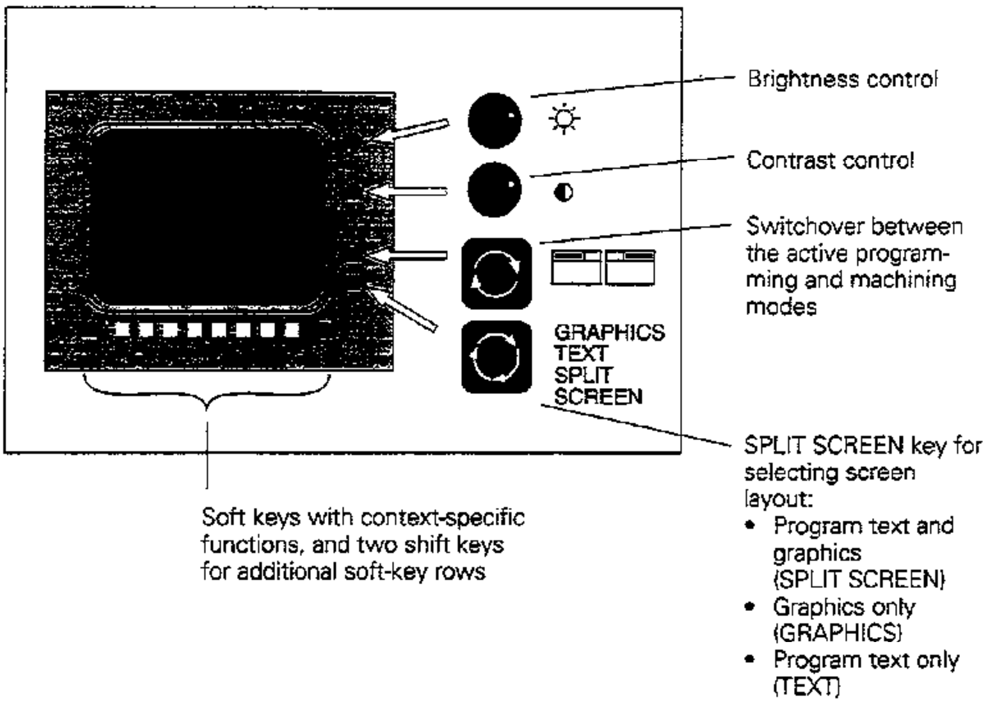
The keys on the TNC keyboard are marked with symbols and abbreviations that make them easy to remember. They are grouped according to the following functions:



The functions of the individual keys are described in the front-cover fold-out.

Machine panel buttons, e.g. I (NC start), are describe in the manual for your machine tool. In the present manual they are shown in gray.

Visual display unit



Headline

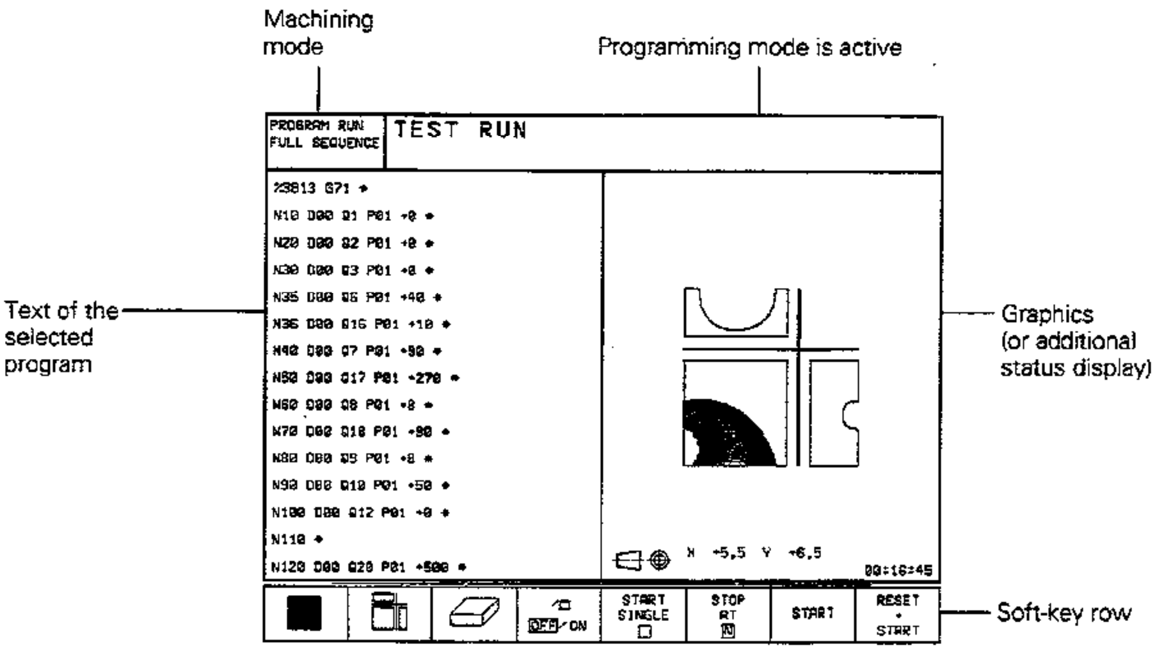
The two selected TNC modes are shown in the screen headline: the machining mode to the left and the programming mode to the right. The currently active mode is displayed in the larger box, where dialog prompts and TNC messages also appear.

Soft keys

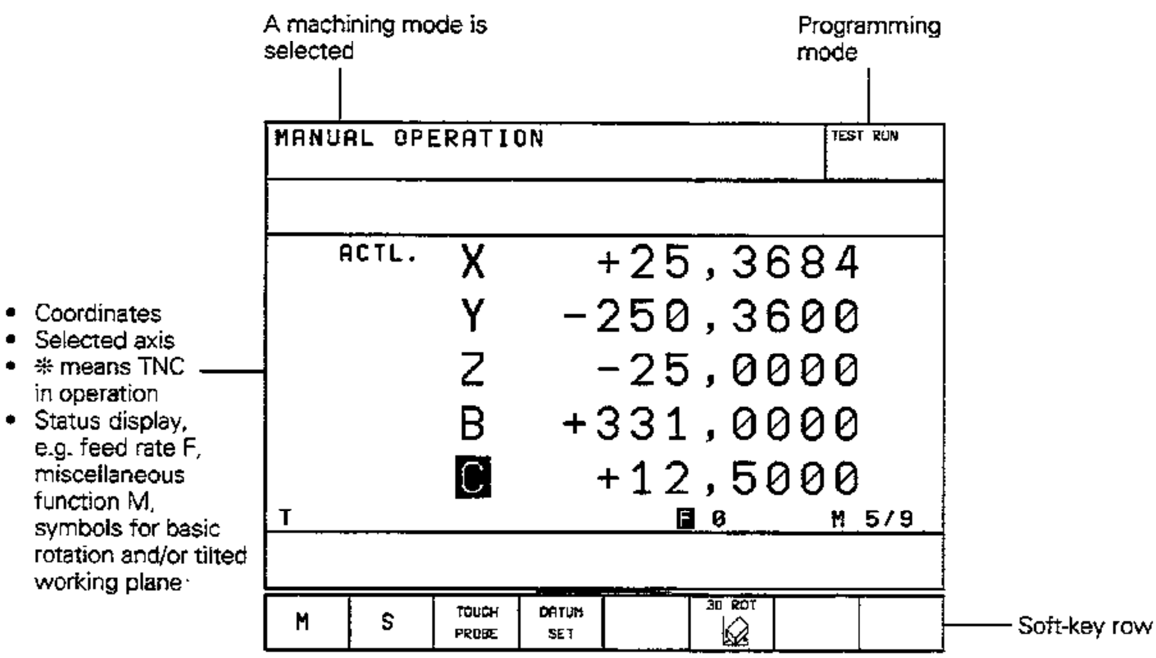
The soft keys select the functions shown in the soft-key row immediately above them. The shift keys to the right and left call up additional soft-key rows. Colored lines above the soft-key row indicate the number of available rows. The line representing the active row is highlighted.

Screen layout of modes

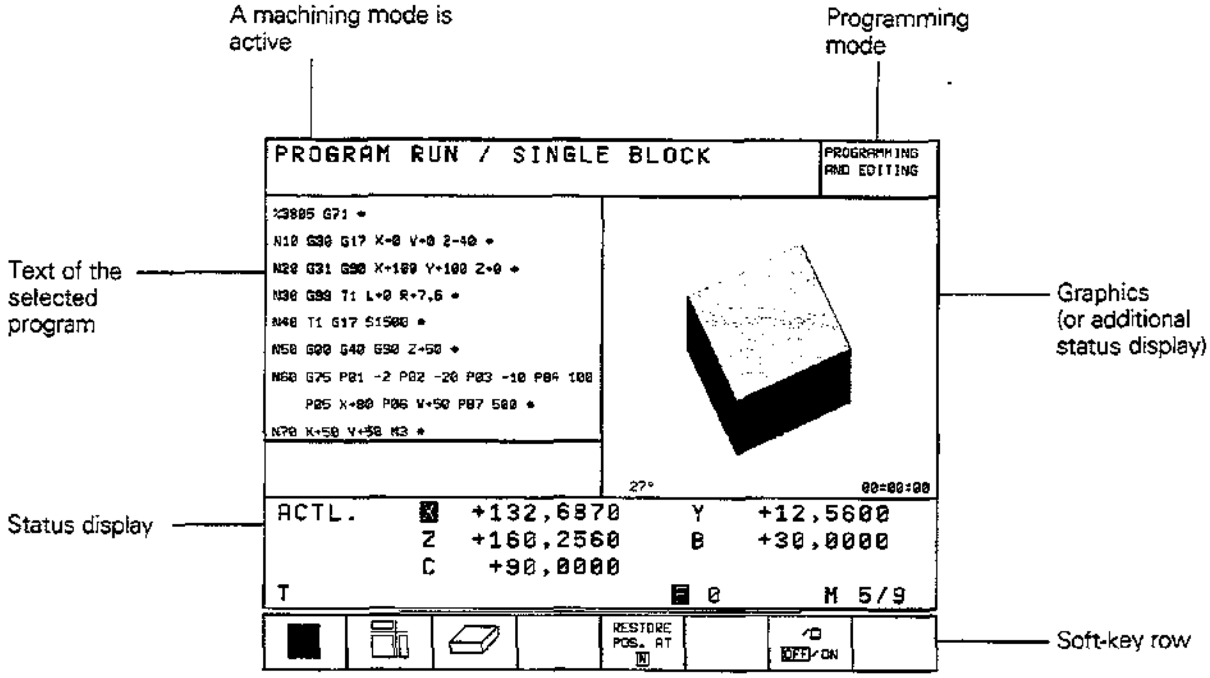
Programming mode:



MANUAL OPERATION and ELECTRONIC HANDWHEEL modes:



PROGRAM RUN operating modes



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)

The TS 120 transmits its signals over cable, while the TS 510 uses infrared light.

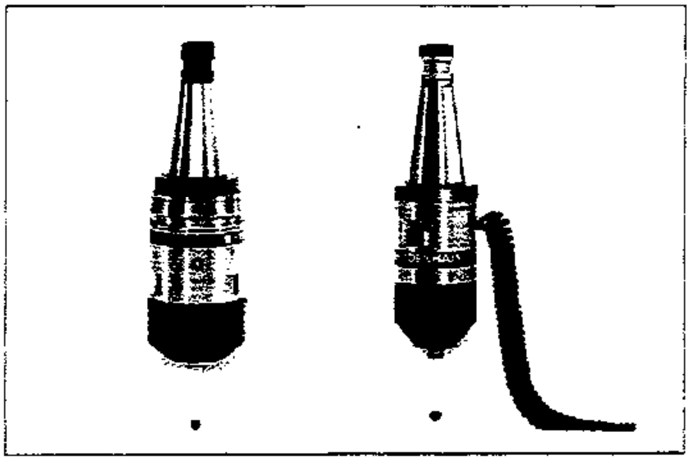


Fig. 1.6: HEIDENHAIN 3D Touch Probe Systems TS 511 and TS 120

Floppy Disk Unit

The HEIDENHAIN FE 401 floppy disk unit enables you to store programs and tables on diskette. It is also a means of transferring programs created on a PC.

Very large programs that exceed the storage capacity of the TNC can be "drip fed": the machine executes each transferred block and erases it immediately, freeing up memory for the next block from the FE.

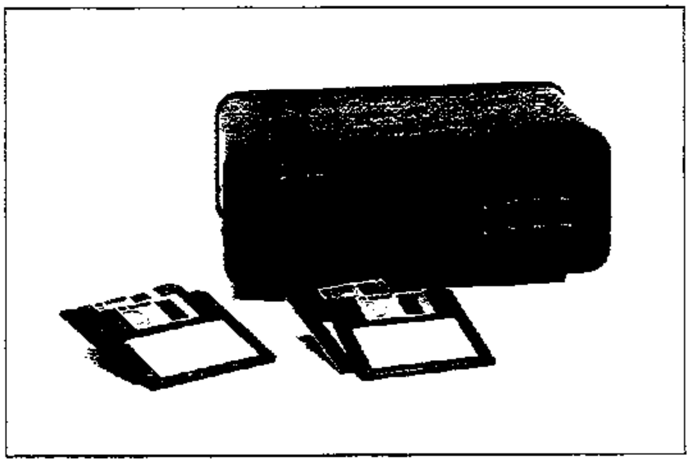


Fig. 1.7: HEIDENHAIN FE 401 Floppy Disk Unit

Electronic Handwheel

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.

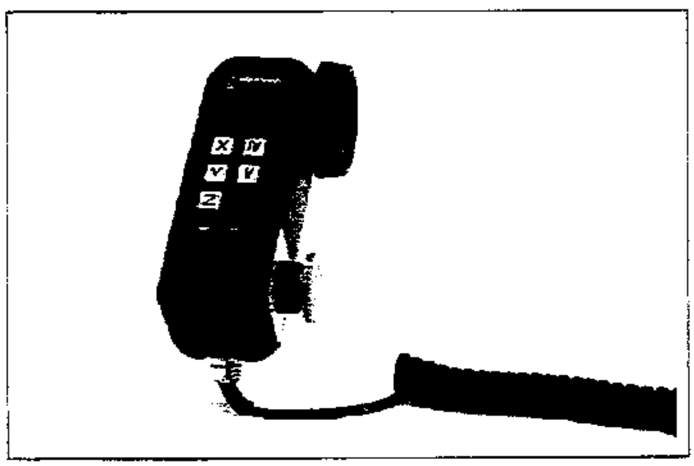


Fig. 1.8: The HR 330 Electronic Handwheel

1.2 Fundamentals of Numerical Control (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.

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.

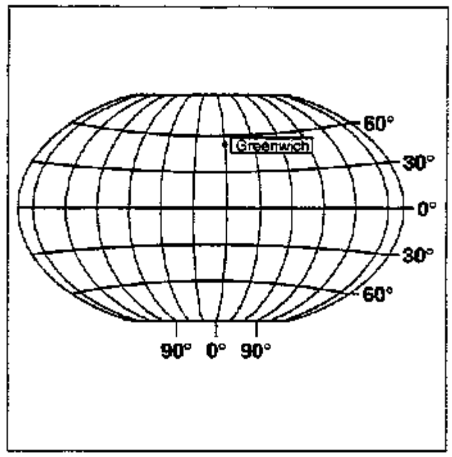


Fig. 1.9: The geographic coordinate system is an absolute reference system

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.

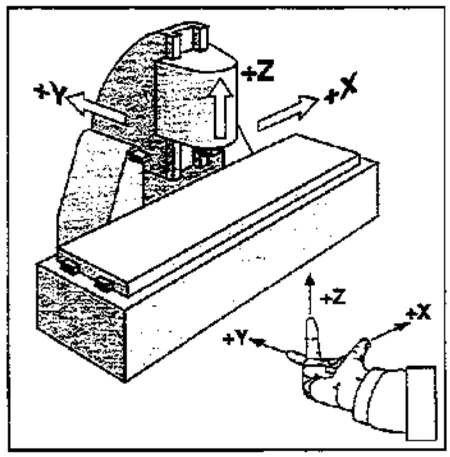


Fig. 1.10: Designations and directions of the axes on a milling machine

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 **A**, **B** and **C**.

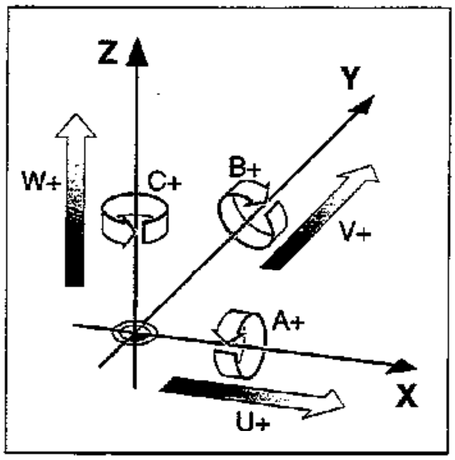


Fig. 1.11: Direction and designation of additional axes

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.

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.

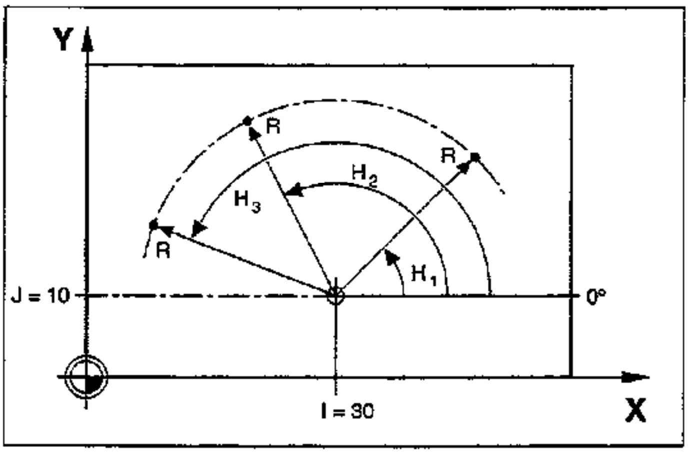


Fig. 1.12: Identifying positions on a circular arc with polar coordinates

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	Reference axis of the angle
I, J	+X
J, K	+Y
K, I	+Z

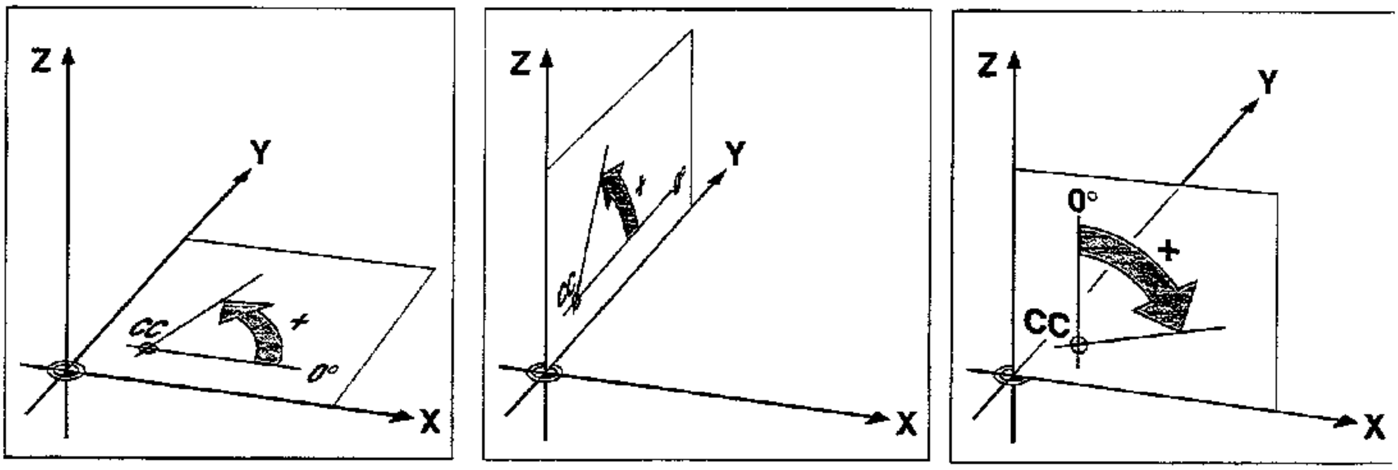


Fig. 1.13: Polar coordinates and their associated reference axes

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 systems. The workpiece, which is aligned with the machine axes, is moved to a certain position relative to the tool and the display is set either to zero or to another appropriate value (e.g., to compensate the tool radius).

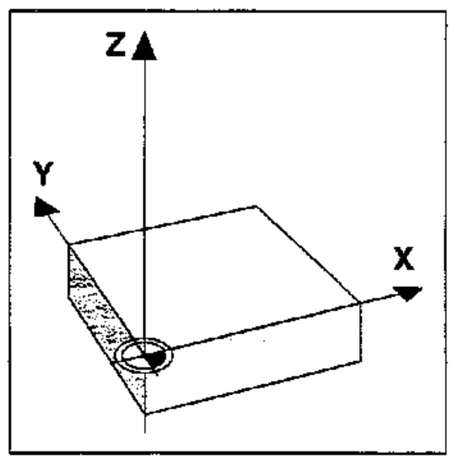
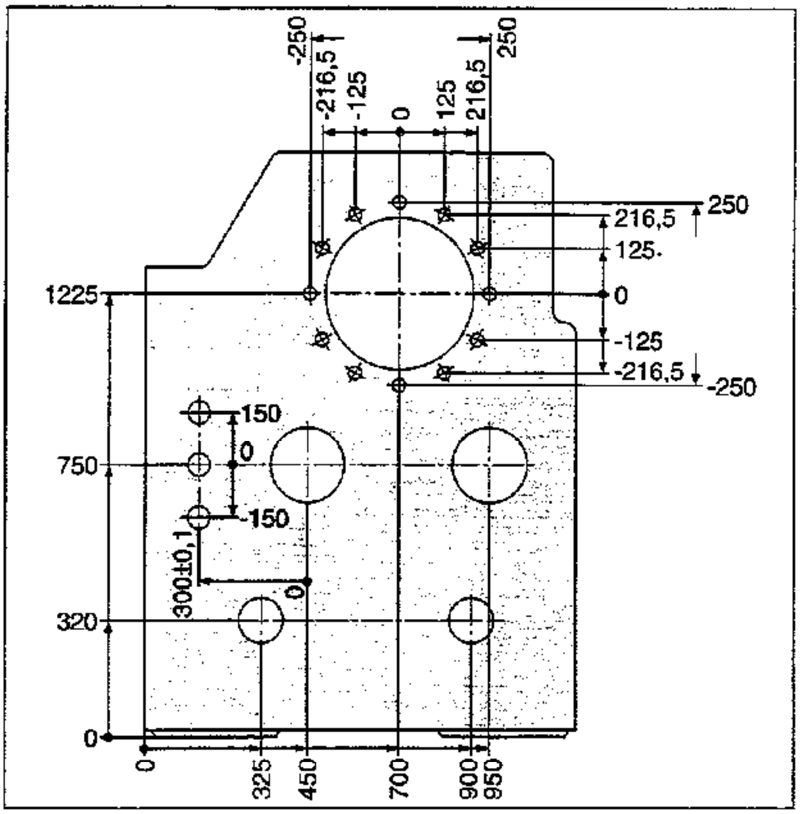


Fig. 1.14: The workpiece datum represents the origin of the Cartesian coordinate system

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.

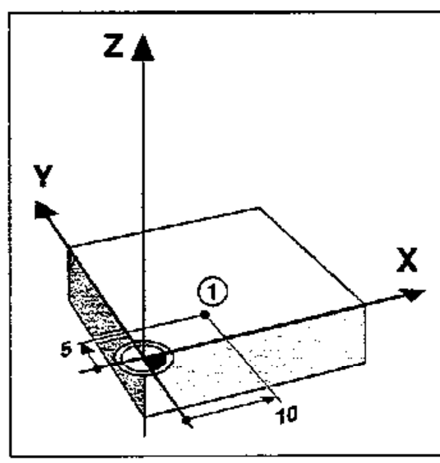


Fig. 1.15: Point ① defines the coordinate system

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.

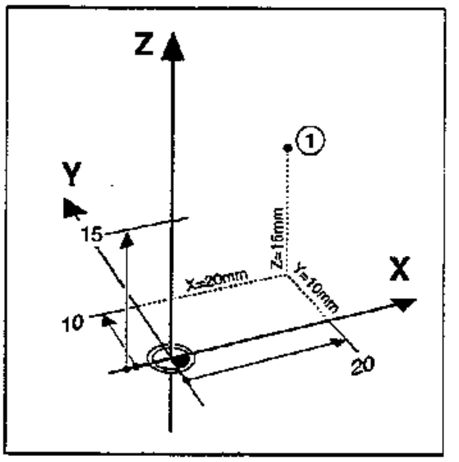


Fig. 1.16: Position definition through absolute coordinates

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.

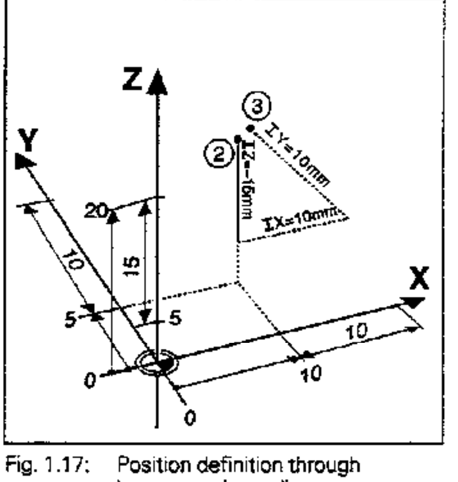


Fig. 1.17: Position definition through incremental coordinates

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.

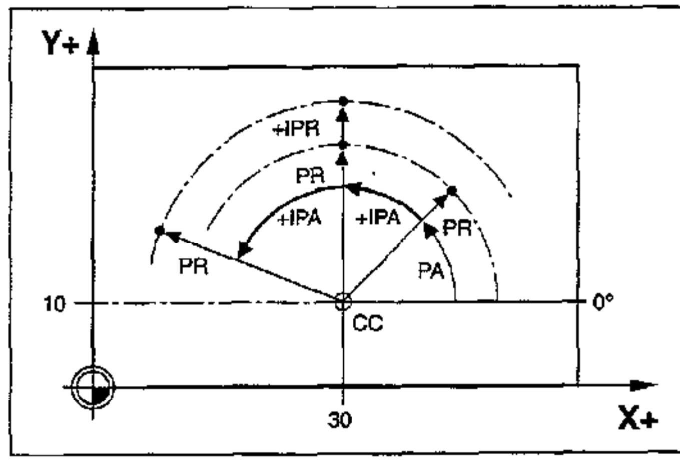
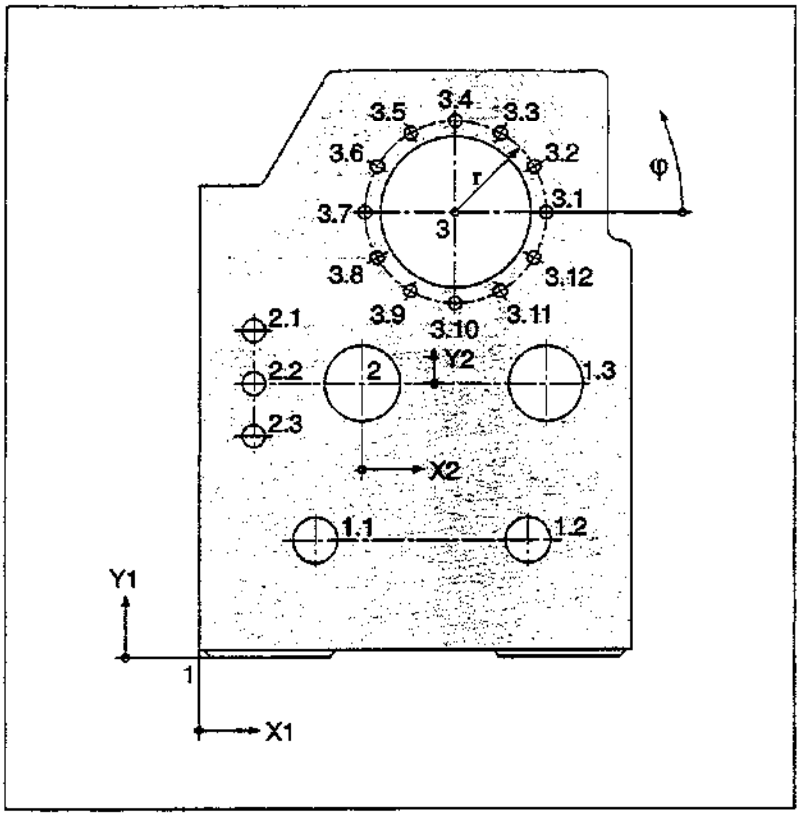


Fig. 1.18: Incremental dimensions in polar coordinates (designated by G91)

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.

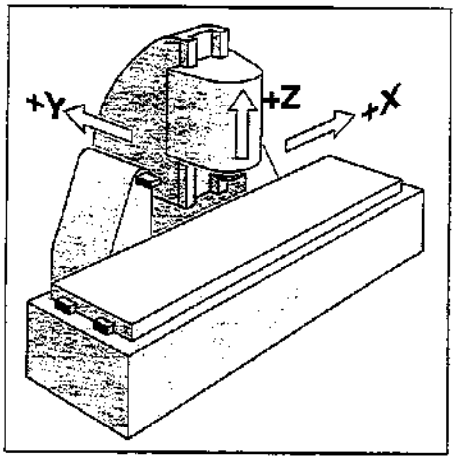


Fig. 1.19: On this machine the tool moves in the Y and Z axes, and the table moves in the +X' axis.

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.

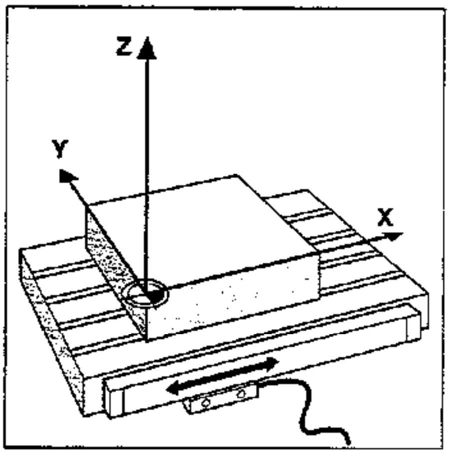


Fig. 1.20: Linear position encoder, here for the X axis

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.

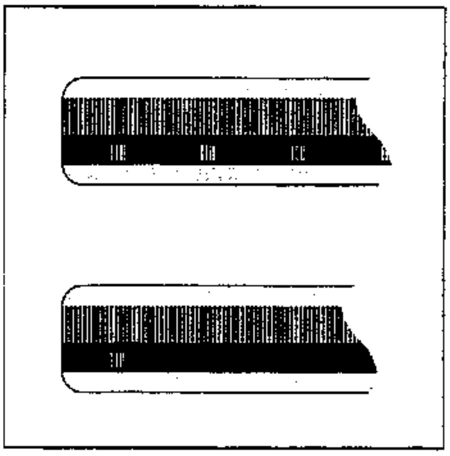


Fig. 1.21: Linear scales: with distance-coded reference marks *(upper illustration)* and one reference mark *(lower illustration)*

1.3 Switch-On

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

MEMORY TEST

The TNC memory is automatically checked.

POWER INTERRUPTED

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

TRANSLATE PLC PROGRAM

The PLC program of the TNC is translated automatically.

RELAY EXT. DC VOLTAGE MISSING

Switch on the control voltage.
The TNC checks the EMERGENCY OFF circuit. ▶ I

MANUAL OPERATION

TRAVERSE REFERENCE POINTS

Move the axes over the reference marks in the displayed sequence:
For each axis press the START key. ▶ I

Cross the reference points in any sequence:
Press the machine axis direction button for each axis
until the reference point has been traversed. ▶ X, Y, ...

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 ANDEDITING 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.

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 and TNC 425, 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.

Graphics during program run

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.

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.

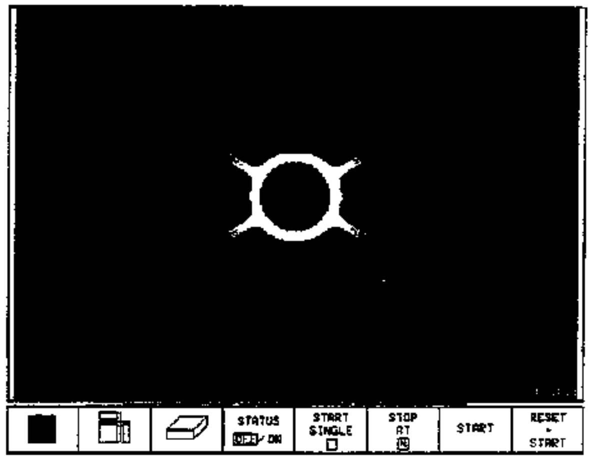
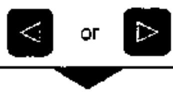
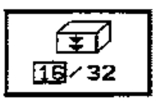


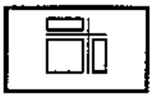
Fig. 1.22: TNC graphics, plan view





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 1-24).

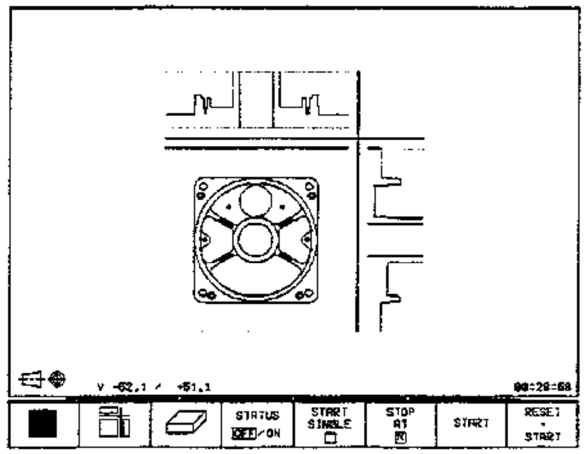


Fig. 1.23: TNC graphics, projection in three planes

Shifting planes

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

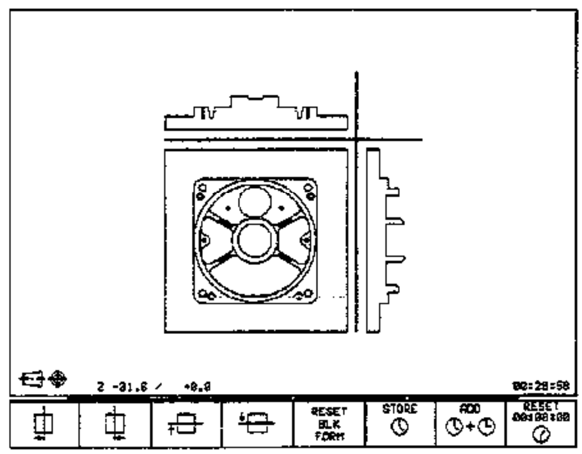
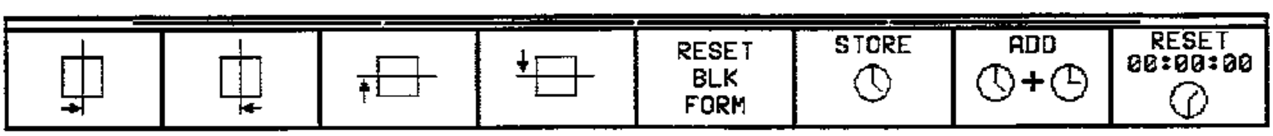
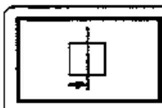
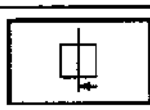
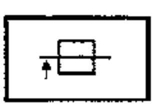
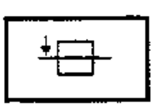


Fig. 1.24: Shifting sectional planes

 or 	Shift the soft-key row.



 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.

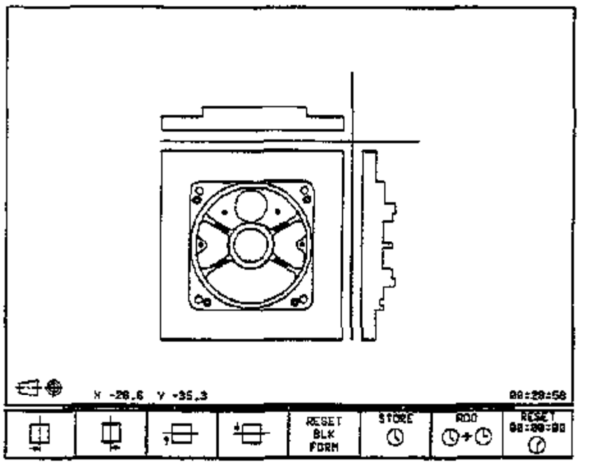
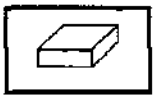


Fig. 1.25: The coordinates of the cursor position are displayed to the lower left of the graphic

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.

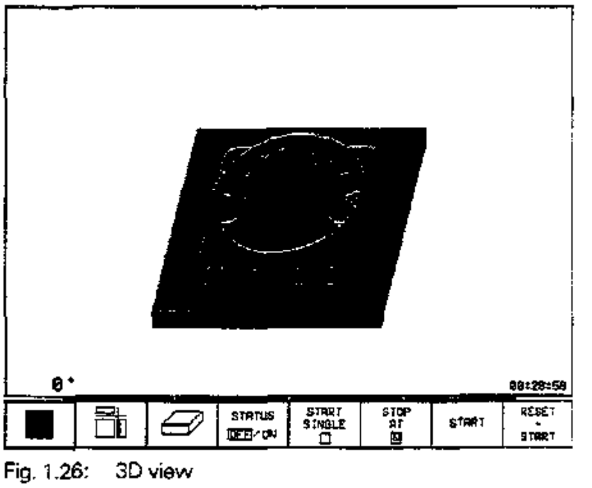
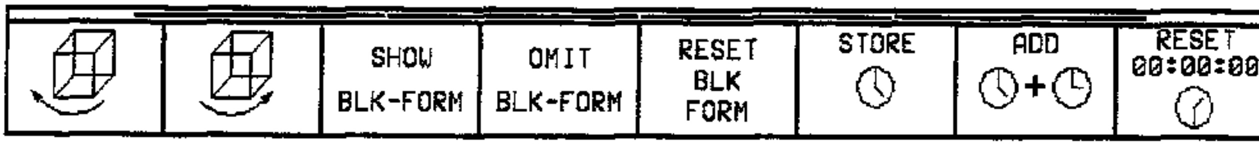
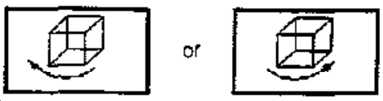


Fig. 1.26: 3D view

To rotate the 3D view:

◁ or ▷	Shift the soft-key row.



	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.

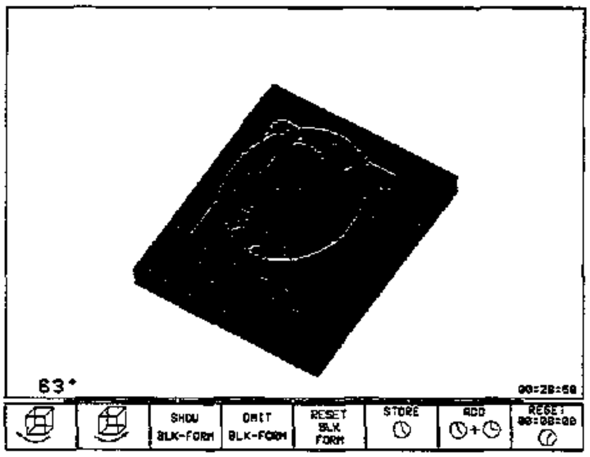


Fig. 1.27: Rotated 3D view

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.

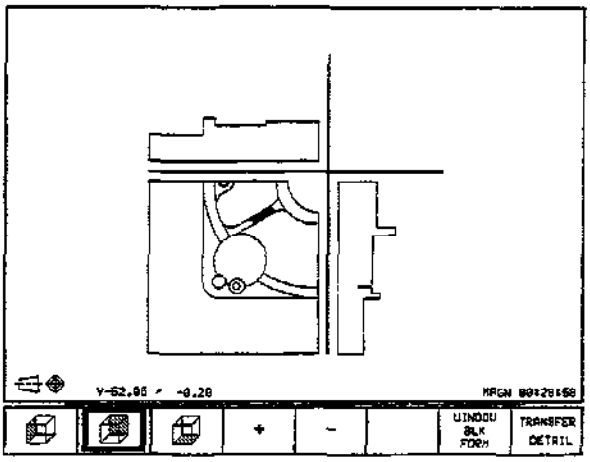


Fig. 1.28: Magnifying a detail of a projection in three planes

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 in 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.

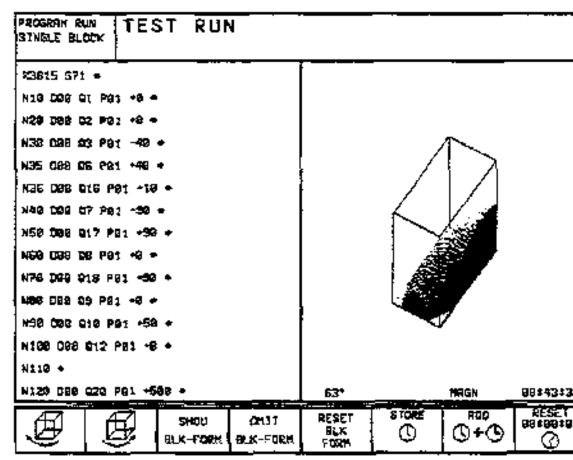


Fig. 1.29: The calculated machining time is shown at the lower right of the workpiece graphic

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 function	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:

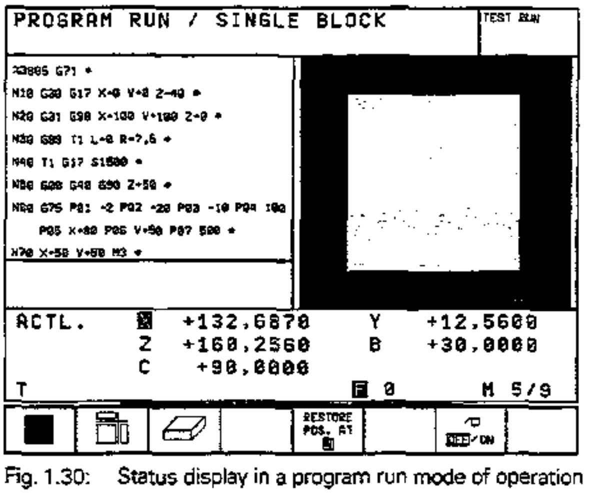


Fig. 1.30: Status display in a program run mode of operation

Additional status displays

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

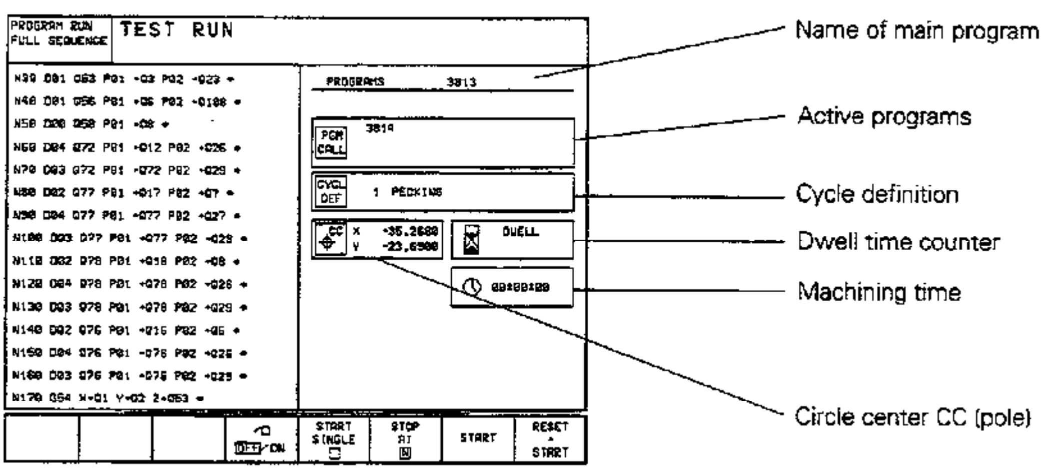
To select additional status displays:

STATUS OFF/ON	Set the STATUS soft key to ON.
◁ or ▷	Shift the soft-key row.

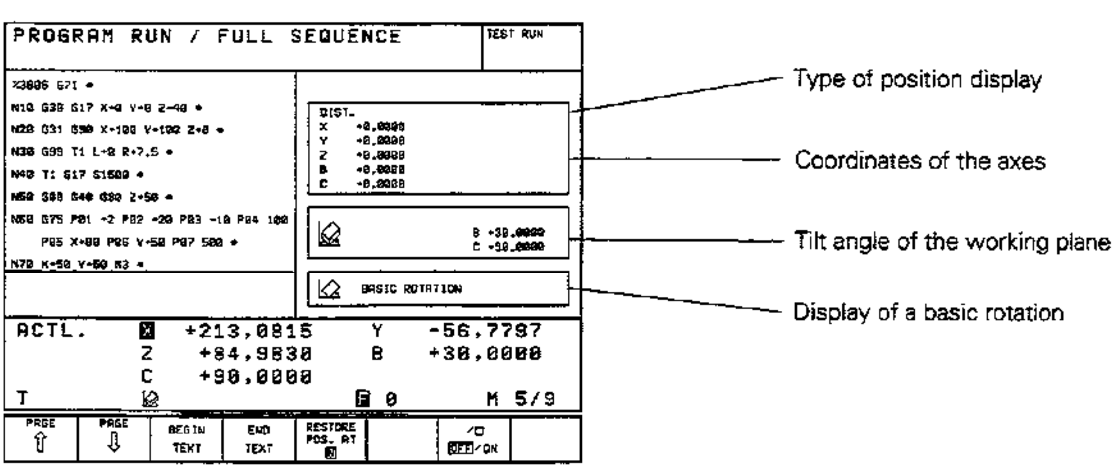
STATUS PGM	STATUS POS.	STATUS TOOL	STATUS COORD. TRANSF.		STORE	ADD	RESET 00:00:00

Additional status display	Soft key
• General program information	STATUS PGM
• Positions and coordinates	STATUS POS.
• Tool information	STATUS TOOL
• Coordinate transformations	STATUS COORD. TRANSF.

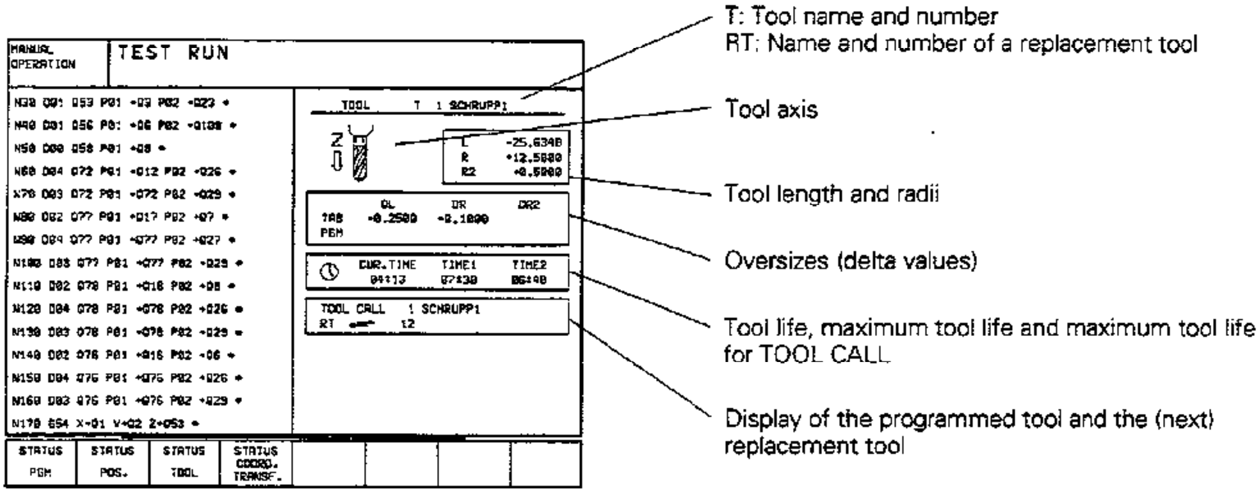
General program information



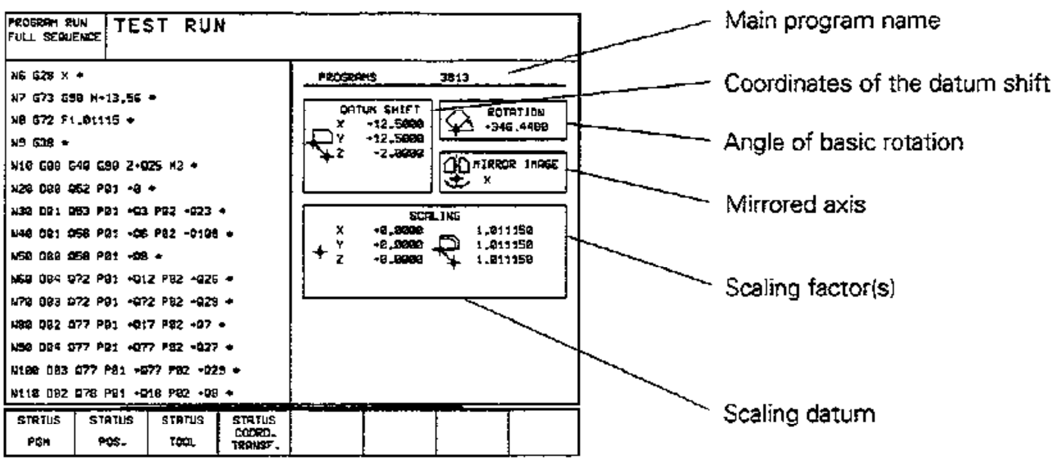
Positions and coordinates



Tool information



Coordinate transformations



1.5 Files

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 • in ISO format	 .H .I
Tables for • Tools • Pallets • Datums	 .T .P .D
Texts as • ASCII files	 .A

Fig. 1.35: 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 9).

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

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. 1.36: File management functions

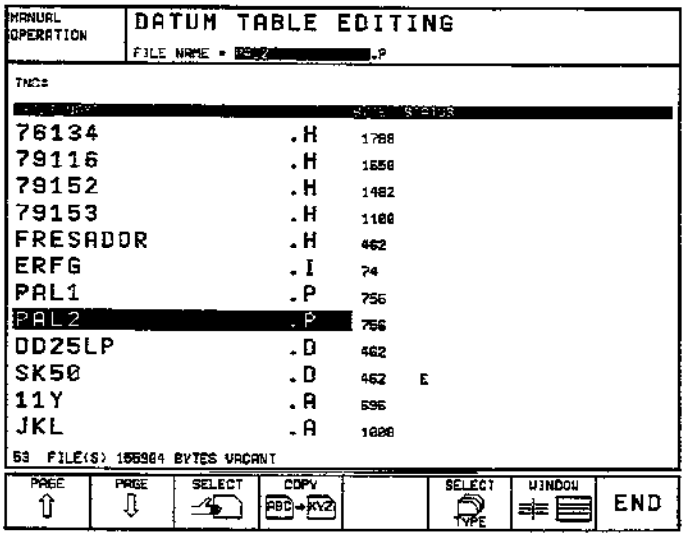


Fig. 1.37: Files are sorted alphabetically and according to type

Select the file directory:

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

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

PGM NAME	Call the file directory.

PAGE ⇧	PAGE ⇩	SELECT	COPY ABC⇒XYZ		SELECT TYPE	WINDOW	END

Initially only HEIDENHAIN dialog (type .H) files are shown. Other files are shown via soft key:

SELECT TYPE	Select the file type.

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

SHOW ALL	Show all files.

You select a file by moving the highlight bar:

Function	Key / Soft key
• Move the highlight bar vertically to the desired file	↓ / ↑
• Move pagewise down/up through the file directory	PAGE ⇩ / PAGE ⇧
• Select the highlighted file	SELECT

To copy a file:

Mode of operation: PROGRAMMING AND EDITING.

PGM NAME	Call the file directory.

Move the highlight bar to the file you wish to copy, for example a file of type .I

COPY ABC ⇒ XYZ	Select the copying function.

DESTINATION FILE = I

Type the new file name into the highlight bar in the screen headline. The file type here is .I

ENT	Copy the file.

To erase a file:

You can erase files in the PROGRAMMING AND EDITING operating mode.

CL PGM	Call the file directory with CL PGM.

PAGE ⇧	PAGE ⇩	DELETE	COPY ABC ⇒ XYZ		SELECT TYPE	WINDOW	END

Move the highlight to the file you wish to delete.

DELETE	Erase the file.

Protected files

A protected file (status P) cannot be erased. If you are sure you wish to erase such a file, you must first remove the protection (see page 1-32).

Protecting, renaming and converting files

In the PROGRAMMING AND EDITING operating mode you can:

- convert files from one type to another
- rename files
- protect files against editing and erasure

PGM NAME	First call the program directory.
>	Then switch the soft-key row.

PAGE ⇧	PAGE ⇩	PROTECT	UNPROTECT	RENAME ABC = XYZ	CONVERT ABC ⇒ XYZ		END

To protect a file:

Move the highlight to the file that you wish to protect.	
PROTECT	Select PROTECT. The file now has status P and cannot be accidentally changed or erased. The protected file is displayed in bright characters.

To cancel file protection:

Move the highlight to the file with status P whose protection you wish to remove.	
UNPROTECT	Select UNPROTECT.
8 6 3 5 7	Type the code number 86357 into the highlight bar in the screen headline.
ENT	Cancel the file protection. The file no longer has the status P.

You can now unprotect further files simply by marking them and pressing the UNPROTECT soft key.

To rename a file:

Move the highlight to the file that you wish to rename.

RENAME ABC = XYZ	Select RENAME.

DESTINATION FILE = .I

Type the new file name into the highlight in the screen headline. The file type cannot be changed.

ENT	Rename the file.

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.

Move the highlight to the file you wish to convert.

CONVERT ABC → XYZ	Select CONVERT.
CONVERT → .A	Select the new file type, here an ASCII text file (type .A).

DESTINATION FILE = .A

Type the new file name into the highlight bar in the screen headline.

ENT	Convert the file.

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.

EXT	Call the program directory for external files.

PAGE ⇧	PAGE ⇩	TRANSFER TNC ⇨ EXT	TRANSFER TNC ⇨ EXT	TRANSFER TNC ? ⇨ EXT	SELECT TYPE	WINDOW	END

→	Move the highlight to the right onto the external file.
WINDOW	Select one-window mode.

PAGE ⇧	PAGE ⇩	DELETE			SELECT TYPE	WINDOW	END

To erase a file on the FE 401B:

↓ / ↑	Move the highlight to the unwanted file.
DELETE	Erase the file in the highlight.

To protect or unprotect a file on the FE 401B:

>	Switch to the next soft-key row.

PAGE ⇧	PAGE ⇩	PROTECT	UNPROTECT			FMT	END

To protect files, use the PROTECT soft key. To remove file protection, use UNPROTECT. The functions for setting and removing file protection are the same as for files stored in the TNC (see page 1-32).

To format a floppy disk in the FE 401B:

>	Switch to the next soft-key row.

PAGE ⇧	PAGE ⇩	PROTECT	UNPROTECT			FMT	END

FMT	Select the formatting function.

NAME OF DISKETTE =	
e.g. 1 ENT	Enter a name and start formatting with ENT.

To convert and transfer files:

The CONVERT soft key is only available if the selected file is in the memory of the TNC, i.e. if it is displayed on the left side of the screen.

EXT	Call the program directory of the external data medium.

PAGE ⇧	PAGE ⇩	TRANSFER TNC ⇒ EXT	TRANSFER TNC ⇒ EXT	TRANSFER TNC ? ⇒ EXT	SELECT TYPE	WINDOW	END

▷	Switch the soft-key row.

PAGE ⇧	PAGE ⇩	CONVERT ABC ⇒ XYZ					END

CONVERT ABC ⇒ XYZ	Convert the file and save it on the external data medium.
CONVERT → .A	Select the target file type, e.g. .A.

DESTINATION FILE =	
e.g. T B 1 ENT	Enter the new file name and start conversion with ENT.

2 Manual Operation and Setup

2.1 Moving the Machine Axes

Traversing with the machine axis direction buttons

▶ 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.
	To stop the axis, press the machine STOP button.

You can only move one axis at a time with this method.

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 back of handwheel).

Interpolation factor	Traverse in mm per revolution
0	20
1	10
2	5
3	2.5
4	1.25
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

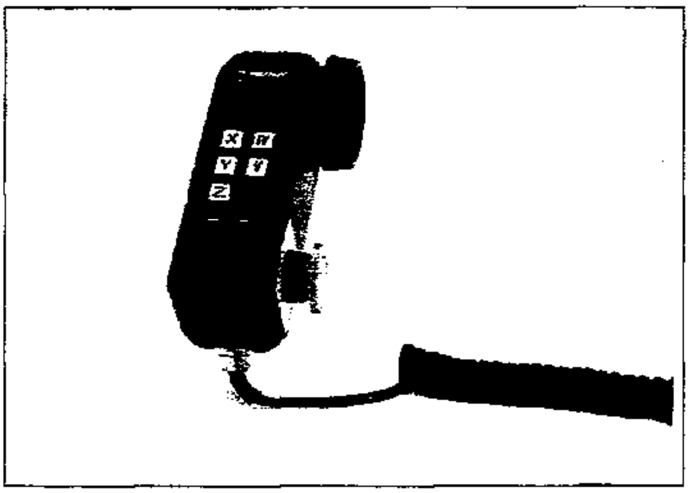


Fig. 2.2: HR 330 electronic handwheel

The smallest programmable interpolation factor depends on the specific machine tool.
It is also possible to move the axes with the handwheel during a program run (see page 5-43).

Using the HR 330 electronic handwheel

Attach the handwheel to a steel surface with the mounting magnets such that it cannot be operated unintentionally.

When you remove the handwheel from its position, be careful not to accidentally press the axis direction keys until the enabling switch is inhibited.

When you hold the handwheel in your hand for machine setup, you must press the enabling switch before you can move the axes with the axis direction keys.

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.

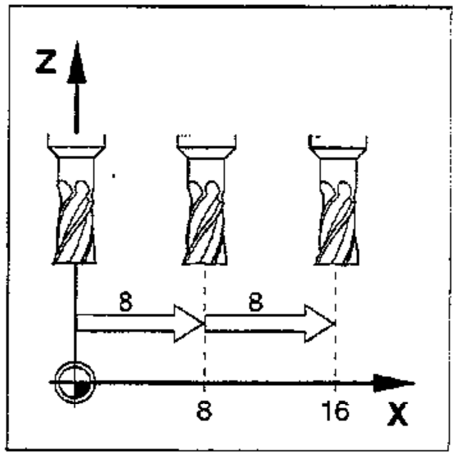


Fig. 2.3: Incremental jog positioning in the X axis

ELECTRONIC HANDWHEEL	
INTERPOLATION FACTOR:	**X = 4**
I	Select incremental jog positioning.

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

- Incremental jog positioning must be enabled by the machine manufacturer.
- The machine manufacturer 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 5-44).

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:

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

These functions are entered directly in a part program in the PROGRAMMING AND EDITING mode.

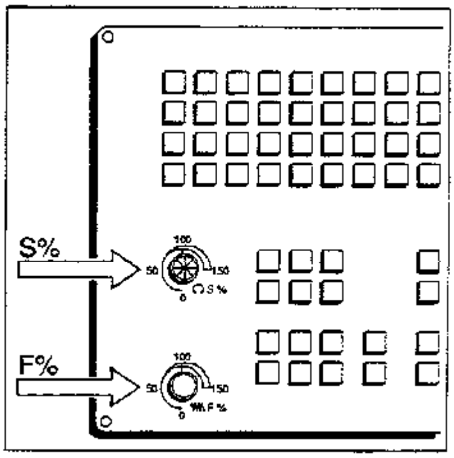


Fig. 2.4: Knobs for spindle speed and feed rate overrides

To enter the spindle speed S:

S	Select S for spindle speed.

SPINDLE SPEED S =

1 0 0 0 ENT	Enter the desired spindle speed (for example, 1000 rpm).
I	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 enter a miscellaneous function M:

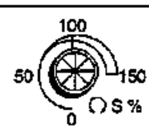
M	Select M for miscellaneous function.

MISCELLANEOUS FUNCTION M =

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

See Chapter 11 for a list of the miscellaneous functions.

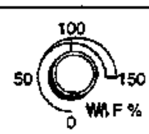
To change the spindle speed S:

	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.

To change the feed rate F:

In the MANUAL OPERATION mode the feed rate is set by a machine parameter.

	Turn the knob for feed rate override. You can vary the feed rate from 0% to 150% of the set value.

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 10-9).

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*.

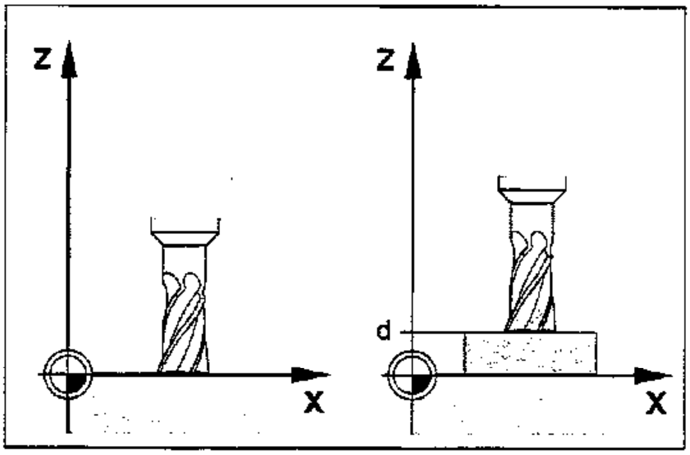


Fig. 2.5: Workpiece setting in the tool axis; *right*, with protective shim

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:

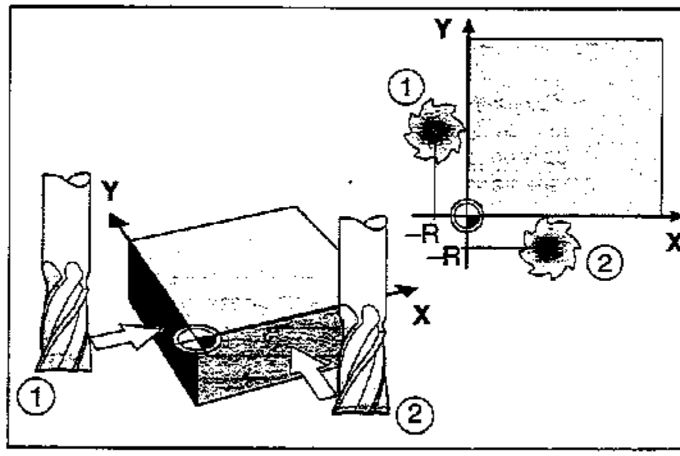


Fig. 2.6: Setting the datum in the working plane; plan view *(upper right)*

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 11-10).

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

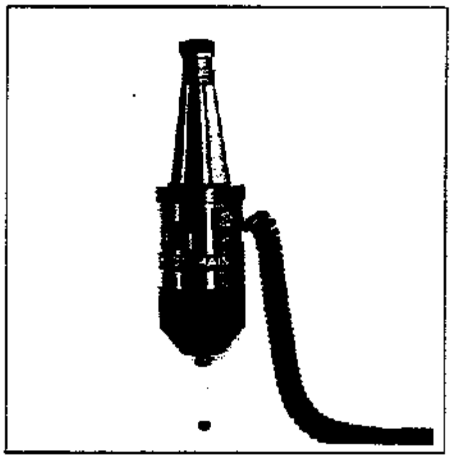


Fig. 2.7: 3D touch probe model TS 120

The TNC must be specially prepared by the machine tool builder 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 11-12).

After you press the machine START button, the touch probe begins executing the selected probing function. The machine manufacturer sets the feed rate F at which the probe approaches the workpiece. 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.

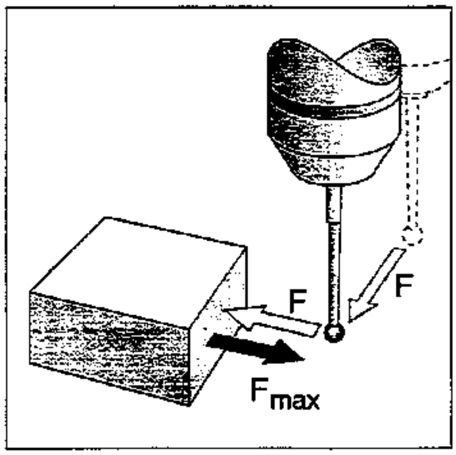


Fig. 2.8: Feed rates during probing

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.

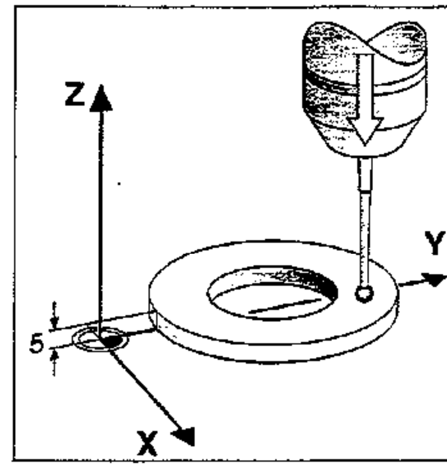


Fig. 2.9: Calibrating the touch probe length

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.

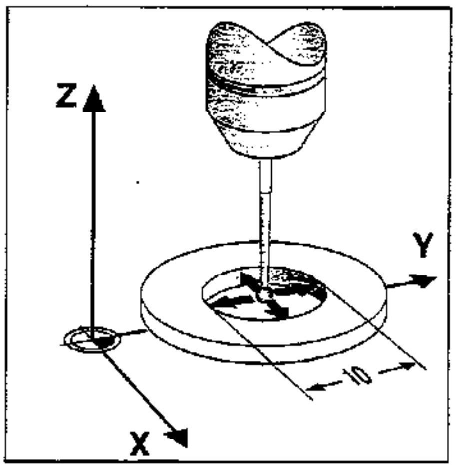


Fig. 2.10: Calibrating the touch probe radius and determining center misalignment

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.

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.

Displaying calibration values

The effective length and radius 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 CALL and CAL R.

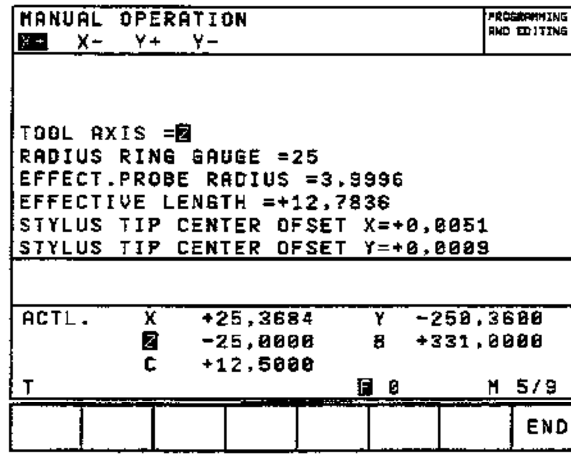


Fig. 2.11: Menu for touch probe radius and center misalignr

Compensating workpiece misalignment

The TNC electronically compensates workpiece misalignment by computing a "basic rotation". You set the rotation angle to the desired angle in respect to the reference axis in the working plane (see page 1-12).

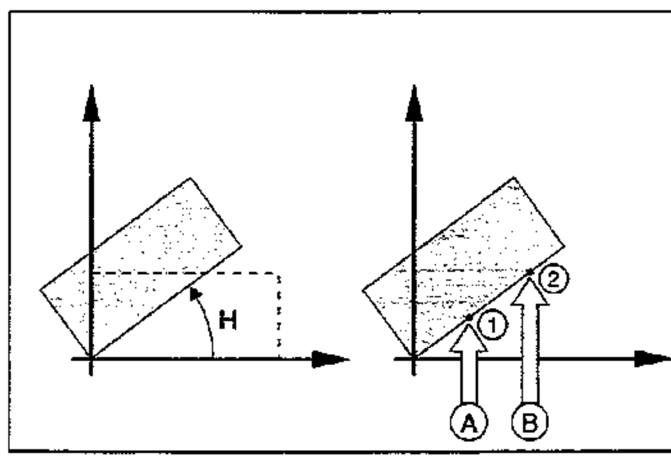


Fig. 2.12: Basic rotation of a workpiece, probing procedure for compensation *(right)*. The broken line is the nominal position, the angle *H* is being compensated.

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 1-22) 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.

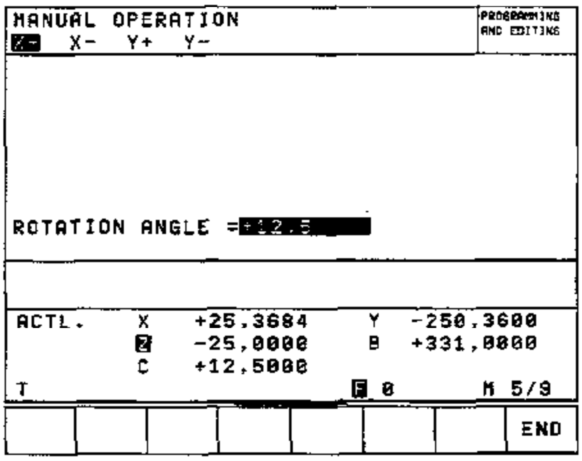


Fig. 2.13: Displaying the angle of an active basic rotation.

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 any axis:

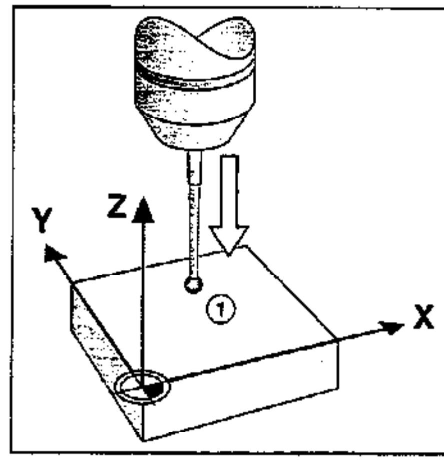


Fig. 2.14: Probing for the datum in the Z axis

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

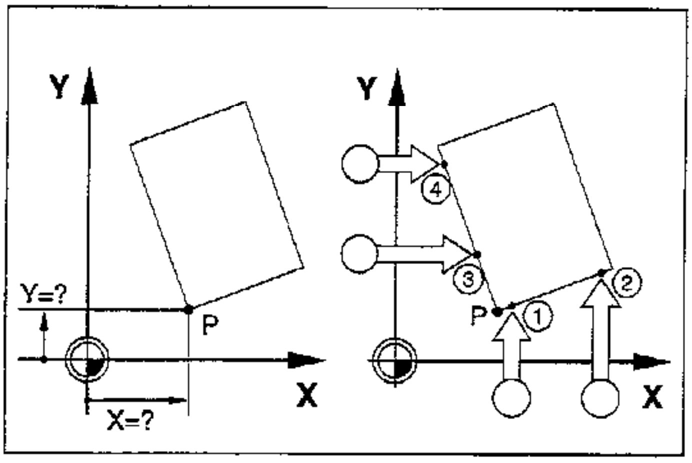


Fig. 2.15: Probing procedure for finding coordinates of corner P

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).

⋮

.
.
.

↓	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.

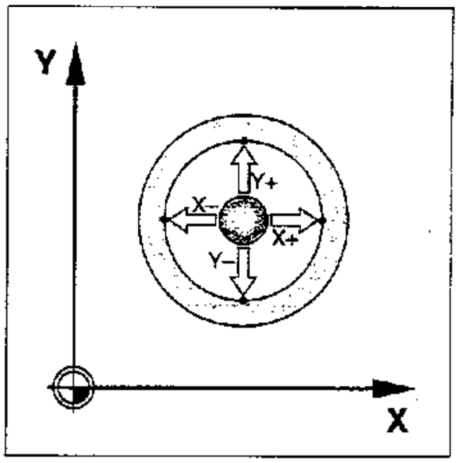


Fig. 2.16: Probing the inside of a cylindrical surface to find the center

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.

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

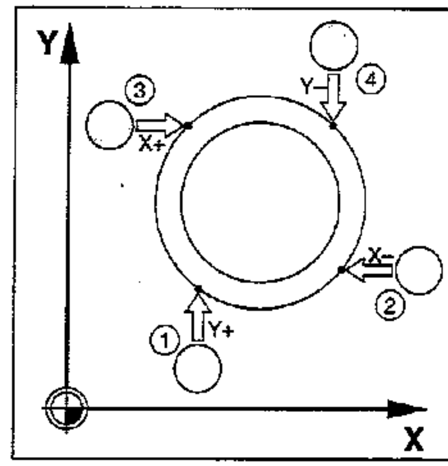


Fig. 2.17: Probing the outside of a cylindrical surface to find the center

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.

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 2-16). 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.

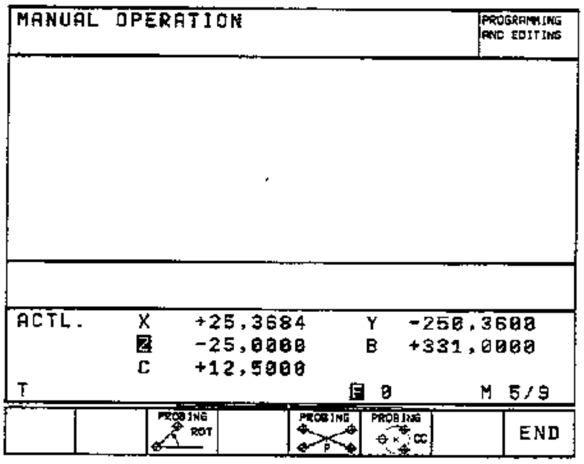


Fig. 2.18: Second soft-key row for TOUCH PROBE

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).	
• **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.	
• **Circle center from 3 holes** The TNC calculates a circle that intersects the centers of all three holes, and finds the center.	

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

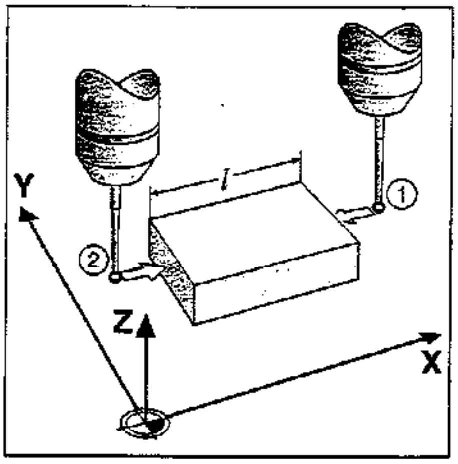


Fig. 2.19: Measuring lengths with the 3D touch probe

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:

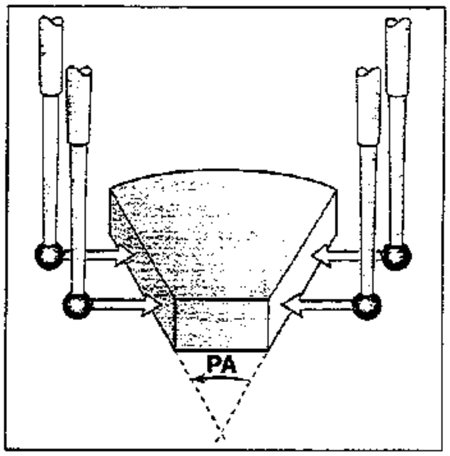


Fig. 2.20: Measuring the angle between two sides of a workpiece

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 (not on TNC 407)

The TNC supports machine tools with swivel heads and/or swivel tables.

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 for this function:

- Oblique holes
- Contours in an oblique plane

The tilting feature is a coordinate transformation. The Z axis remains parallel to the tool axis and the X/Y plane is perpendicular to the tool axis.

On machines with swivel tables the position of the tool axis relative to the machine coordinate system does not change. The coordinate system is not tilted; the slant of the working plane is compensated by tilting the table.

On machines with swivel heads, however, the coordinate system does change. The slant of the working plane is compensated by tilting the coordinate system.

In order to run a program in a tilted plane, the tool must first be pre-positioned in a conventional way – for example with a G00 block.

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 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.

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.
- When combining coordinate transformation cycles, you can use a procedure such as the following to activate them:

  1. Activate datum shift
  2. Activate tilting function
  3. Activate rotation

  Use the reverse procedure for resetting. The cycle that was last defined is reset first, e.g.:

  1. Activate rotation
  2. Activate tilting function
  3. Reset datum shift

The functions for tilting the working plane are interfaced to the TNC and the machine tool by the machine tool builder. He can give you more detailed information on how to enter the individual axes for his machine.

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	Terminate 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.

To reset:

Set TILT WORKING PLANE to INACTIVE.

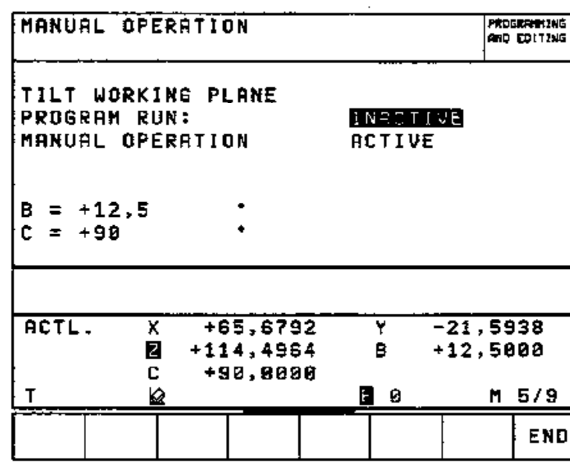


Fig. 2.21: Menu for manual tilting in the MANUAL OPERATION mode

3 Test Run and Program Run

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 data
- Impossible jumps

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:

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.

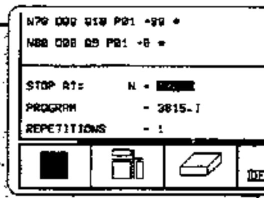
Select the TEST RUN mode and go to the program beginning.

STOP AT N	Select a partial test run.

STOP AT: N =

PROGRAM =

REPETITIONS =



e.g. 5 ENT	Enter the block number N at which you wish the test to stop.
e.g. 1 2 3 ENT	Enter the name of the program that contains the block with block number N.
e.g. 1 ENT	If N is located in a program section repetition, enter the number of repetitions that you wish 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	PAGE ⇩
• Go forward in the program by one screen page	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 part program.
Only in mode PROGRAM RUN / SINGLE BLOCK I for each block	Run each block of the part program separately.

Feed rate and spindle speed can be changed with the override knobs. You can superimpose handwheel positioning onto programmed axis movements during program run (see page 5-43).

Interrupting machining

There are various 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. The part program 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.

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.

Resuming program run after an interruption

When a program run is interrupted, the TNC stores:

- The data of the last tool called
- Active coordinate transformations
- The coordinates of the last defined circle center
- The count of a running program section repetition
- The number of the last CALL LBL block

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

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.

The TNC recalculates these data for resuming program run at a certain block (RESTORE POS AT N).

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

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

With the RESTORE POS AT N feature (block scan) you can start a part program at any desired block. 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.

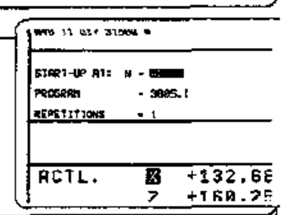
- The RESTORE POS AT N feature must be enabled by the machine tool builder.
- Mid-program startup must not begin in a subprogram.
- All necessary programs, tables and pallet files must be selected in a program run mode of operation.
- If the part program contains a programmed interruption before the 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.

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 =



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, I	Move the axes in the sequence that the TNC proposes on the screen.
RESTORE Z, I RESTORE X, I ...	Move the axes in any sequence.
I	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 the program with/without blocks preceded by a slash.

This function does not work for G99 blocks.

3.4 Blockwise Transfer: Testing and Running Long Programs

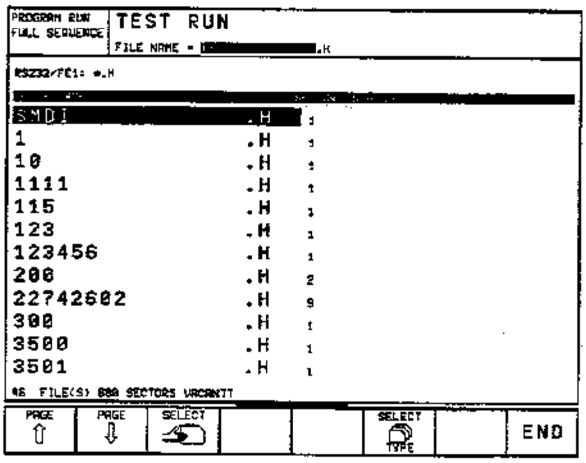


Fig. 3.1: TNC screen during blockwise transfer

Part 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 space 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).

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.

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.

As an alternative, you can call the external program with % EXT (see page 6-8) and perform a mid-program startup (see page 3-8).
You can use machine parameters (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.

4 Programming

4 Programming

In the PROGRAMMING AND EDITING mode of operation (see page 1-25) 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*.

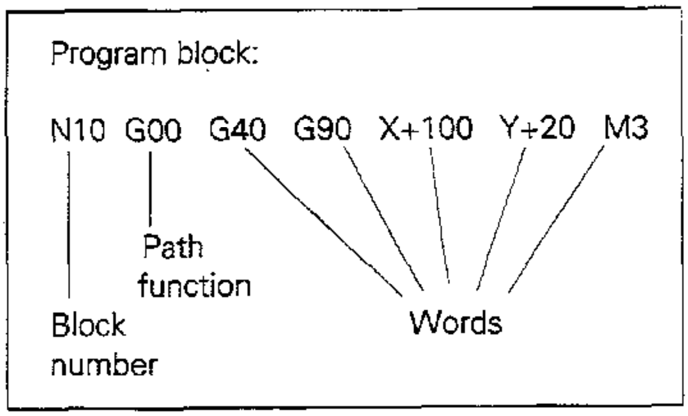


Fig. 4 1: Program blocks consist of words of specific information

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 information, 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.

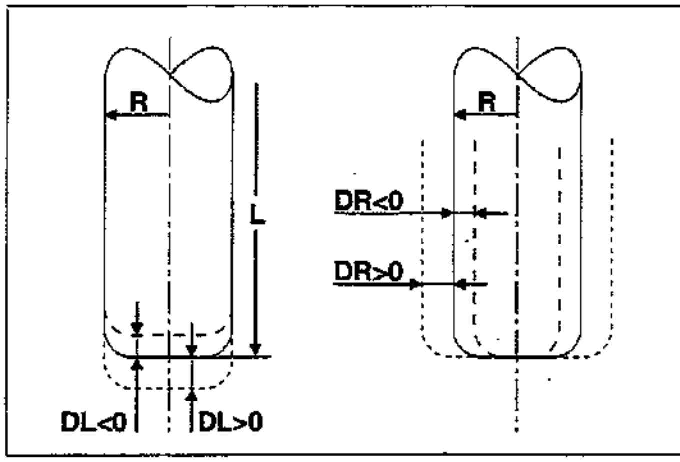


Fig. 4.2: Oversizes DL, DR on a toroid cutter

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

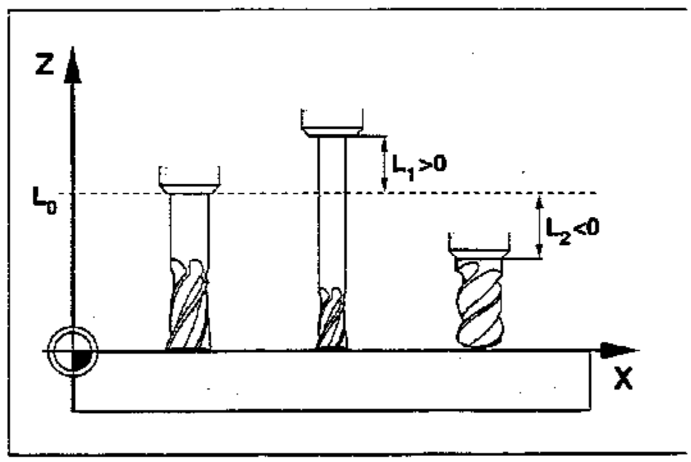


Fig. 4.3: Tool lengths are entered as the difference from the zero tool

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 4-24).

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 4-24).

Entering tool data in tables

A tool table is a file in which the tool data for all tools are stored together. The maximum number of tools per table (0 to 254) is set in machine parameter MP 7260.

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	↑ ↓ ← →
• Go to the beginning/end of the table	BEGIN TABLE / END TABLE
• Go to the next/previous table page	PAGE ⇩ / PAGE ⇧
• Go to the beginning of the next line	NEXT LINE
• Look for the tool name in the tool table	▷ FIND TOOL NAME

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 any tool table other than TOOL.T:

PROGRAMMING AND EDITING	
PGM NAME	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
- Tool name: NAME
- Maximum and current tool life: TIME1, TIME2, CUR.TIME
- Number of a replacement tool: RT
- Tool lock: TL
- Tool comment: DOC

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.

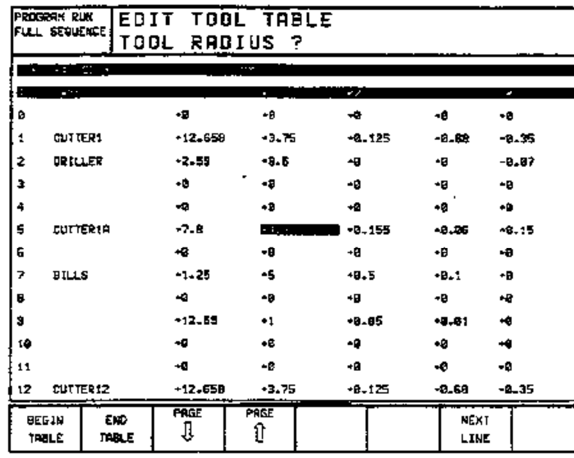


Fig. 4.4: Left part of the tool table

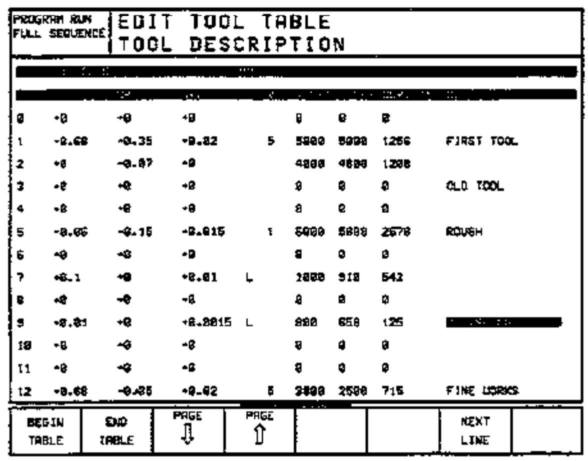


Fig. 4.5: Right part of the tool table

To read-out or read-in a tool table:

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).

See also page 9-2.

Abbreviation	Input	Dialog
T	Number by which the tool is called in the program	–
NAME	Number by which the tool is called in the program (only for conversational programming)	TOOL NAME ?
L	Value for tool length compensation	TOOL LENGTH L ?
R	Tool radius R	TOOL RADIUS R ?
R2	Tool radius R2, for toroid cutter	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 (only for conversational programming)	TOOL RADIUS OVERSIZE 2 ?
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 TIME1.	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)	MAX. 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

Fig. 4.6: 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 also the 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 4-10).

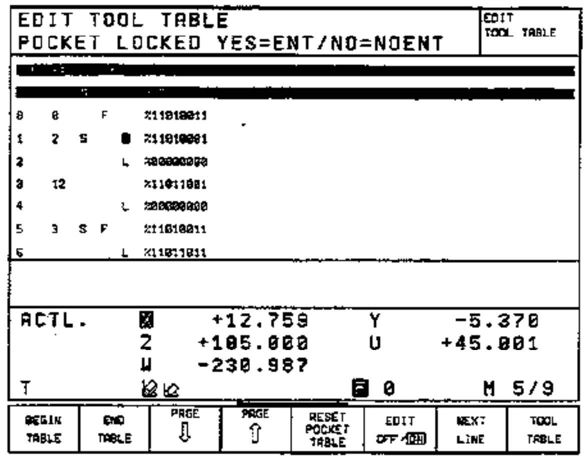


Fig. 4.7: Pocket table for the tool changer

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 T	**P**ocket number of the tool **T**ool number	– TOOL NUMBER
F L	**F**ixed tool number. The tool is always returned to the same pocket. **L**ocked pocket	FIXED POCKET YES = ENT / NO = NOENT POCKET LOCKED YES = ENT / NO = NOENT
ST	**S**pecial **T**ool with large radius requiring several pockets in the tool magazine. Enter the number of pockets to be locked in front of and behind the special tool.	SPECIAL TOOL
PLC	Information on this tool that should be sent to the PLC	PLC STATUS

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

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
- Continue 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 is reduced.

Automatic tool change: M101

Standard behavior – without M101

When the tool reaches the maximum tool life (TIME1), the TNC interrupts program run (depending on the particular machine).

Automatic tool change – with M101

The TNC automatically changes the tool if the tool life (TIME1 or TIME2) expires during program run.

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.

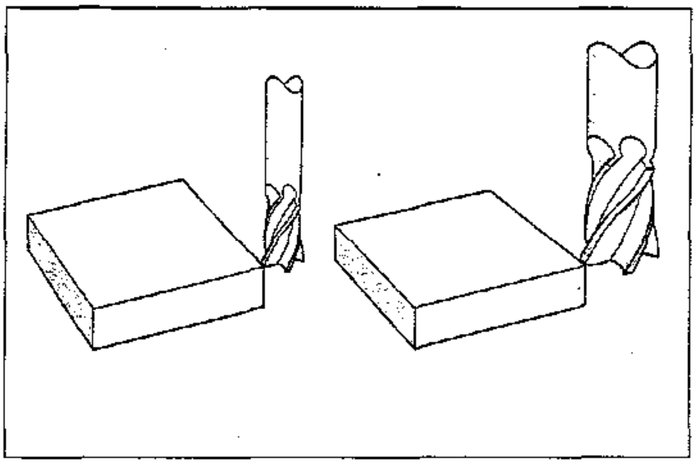


Fig. 4.8: The TNC compensates both the length and radius of the tool

Effect of tool compensation values

Tool length

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

Length compensation is cancelled by calling 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.

Radius compensation is cancelled by programming 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)

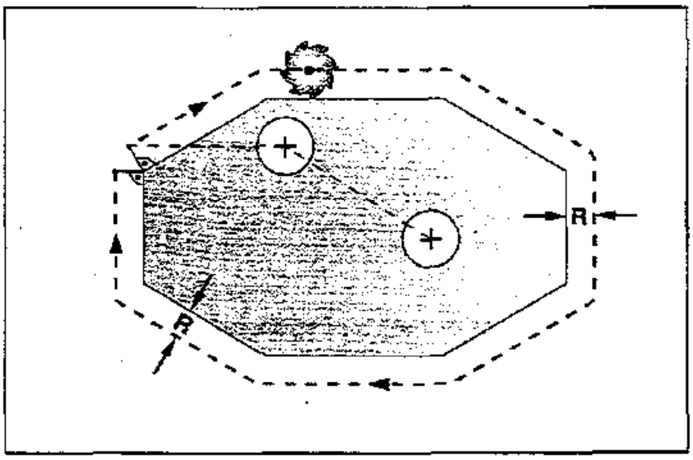


Fig. 4.9: Programmed contour (—, +) and the path of the tool center (- - -)

Movement without radius compensation: G40

The tool center moves to the programmed coordinates.

Applications:

- Drilling and boring
- Pre-positioning

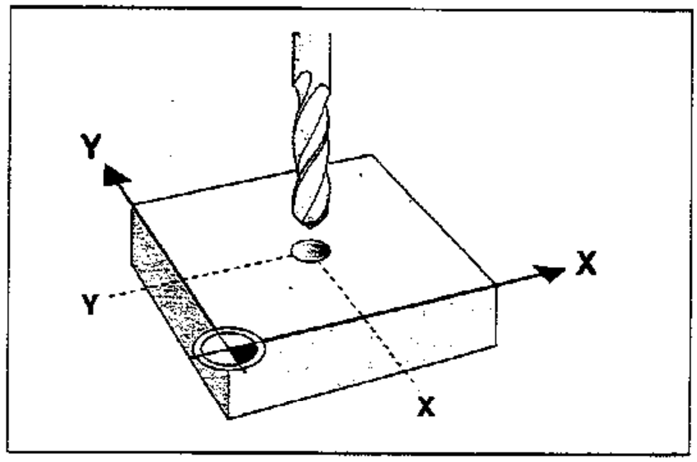


Fig. 4.10: These drilling positions are entered without radius compensation

Tool movement with radius compensation: G41, G42

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.

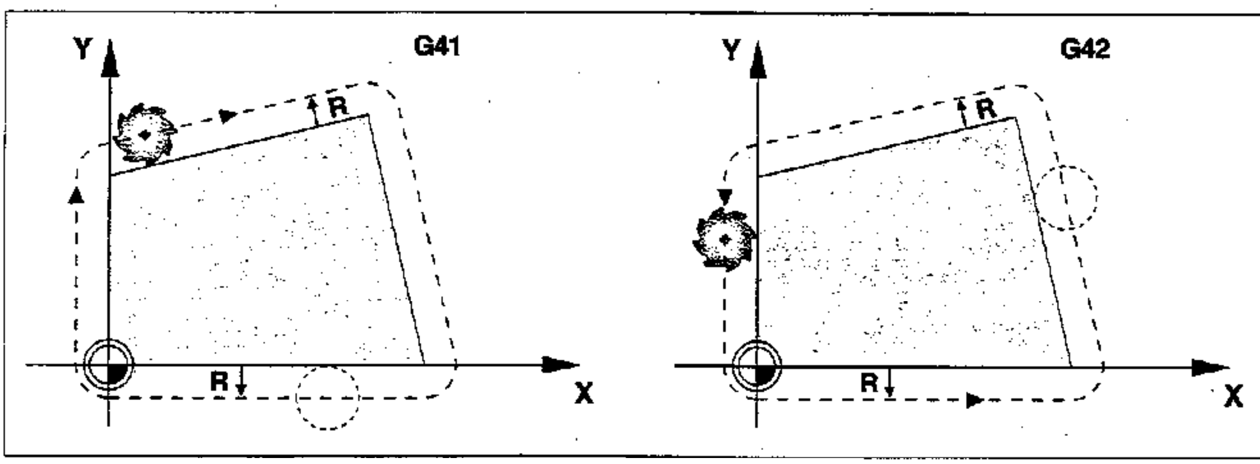


Fig. 4.11: The tool moves to the left *(G41)* or right *(G42)* of the path during milling

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.

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

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.

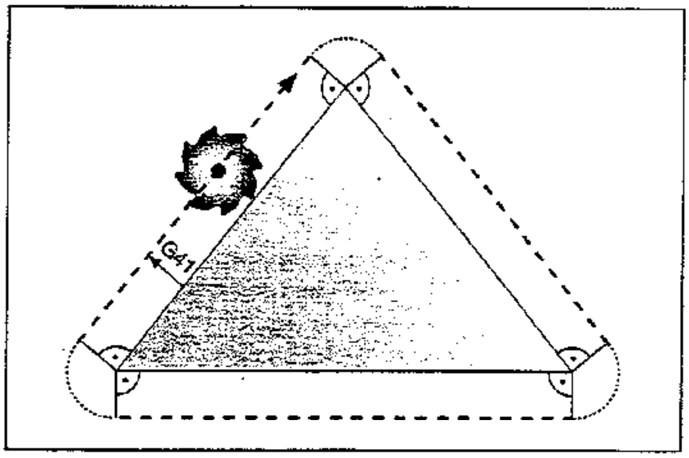


Fig. 4.12: The tool "rolls around" outside corners

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

Inside corners

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.

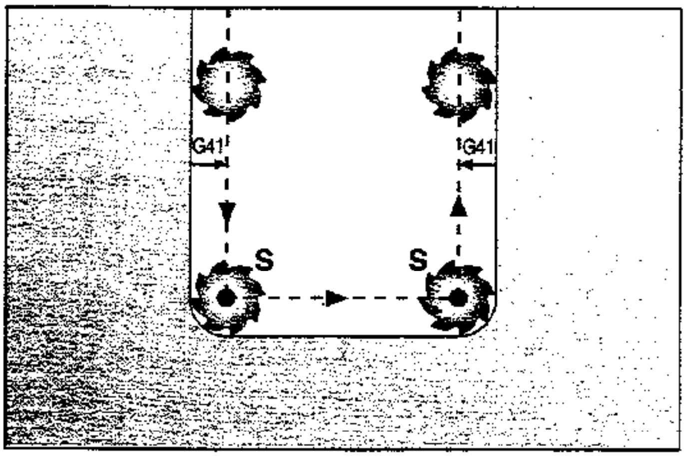


Fig. 4.13: Tool path for inside corners

4.4 Program Initiation

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 mm long.

The dialog for defining the blank form starts automatically at every program initiation. It can also be called with the BLK FORM soft key.

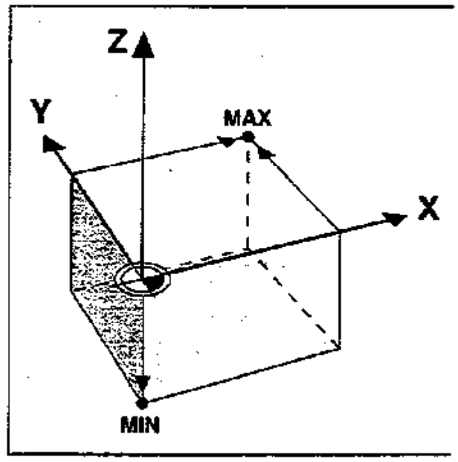


Fig. 4.18: MIN and MAX points define the blank form.

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	Select the file directory

Select any file of type .I, for example OLD .I

FILE NAME = OLD .I

N E W ENT	Enter the name of the new file, for example NEW .I

MM = ENT / INCH = NO ENT

ENT or NO ENT	Indicate whether the dimensions will be entered in millimeters (G71) or inches (G70)

G 3 0	G function for input of the MIN point
G 1 7	Define the tool axis: G17 means 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

The tool-related data can be determined with the aid of diagrams (see page 11-20).

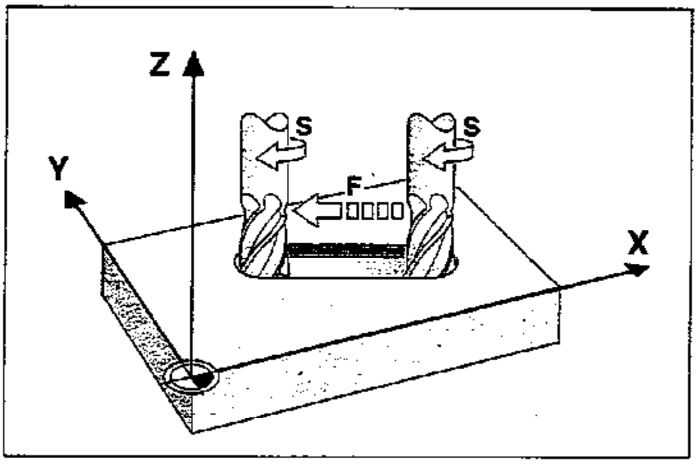


Fig. 4.15: Feed rate F and spindle speed S of the tool

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 30,000 mm/min or 1181 ipm (TNC 425: 300,000 mm/min or 11,811 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 2-5).

Spindle speed S

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

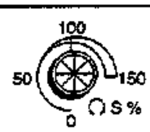
Input range:
S = 0 to 99,999 rpm

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:

	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

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.

Some M functions are not effective on certain machines. The machine tool builder may also add some of his own M functions.

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 8-48).

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.

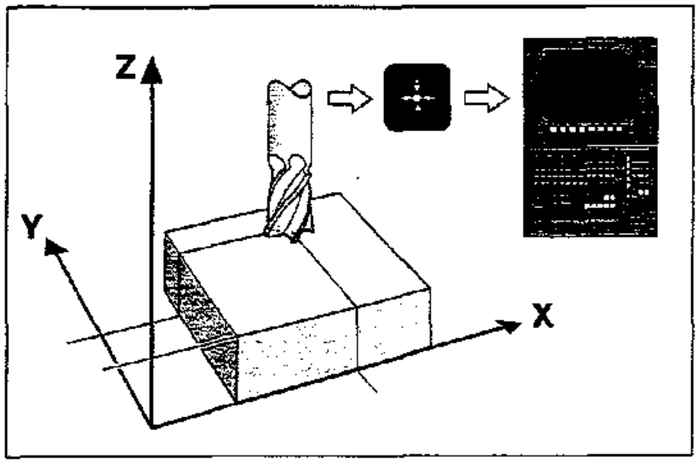


Fig. 4.16: Storing the actual position in the TNC

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 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.

4.9 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 wish to edit other types of files with the text editor, you must first convert them (see page 1-31).

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**

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 highlighted 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 OVERWRITE modes with the soft key at the far left. The selected mode is shown enclosed in a frame.

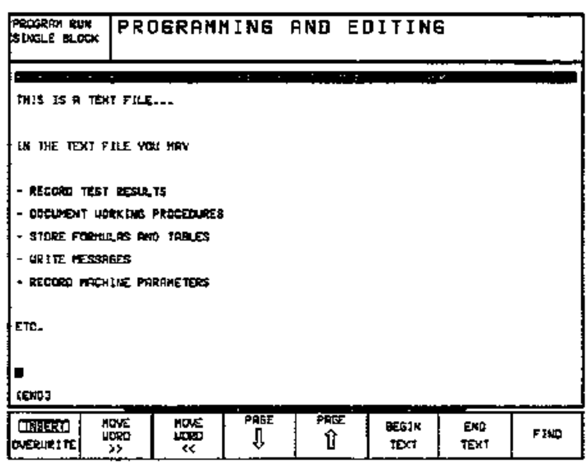


Fig. 4.17: TNC text editor screen

Entering text

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)	⌫
• 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

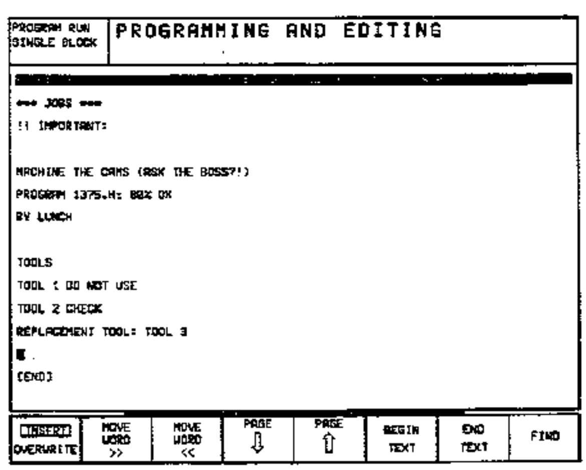


Fig. 4.18: Text editor screen with exercise text

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.
↓	Move the cursor to the end of the block.
REMOVE BLOCK	Erase the text and store temporarily.
↑	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.A.
ENT	Copy into a another file. Text block remains marked.

4.10 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 directory.
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 manufacturer can give you further information on this.

The following functions help you to create and change pallet tables:

Function	Key / Softkey
• Move the highlight	↑ ↓ ← →
• Go to the beginning/end of the table	BEGIN TABLE / END TABLE
• Go to the next/previous page of the table	PAGE ⇩ / PAGE ⇧
• Insert/delete the last line in the table	INSERT LINE / DELETE LINE
• Go to the beginning of the next line	NEXT LINE

4.11 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.

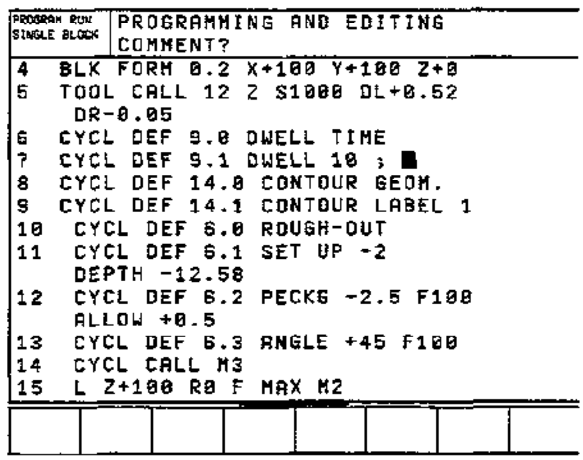


Fig. 4.19: Dialog for entering comments

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
N60    G01 G41 F100 *                     of the block.
.
.
.
```

5 Programming Tool Movements

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).

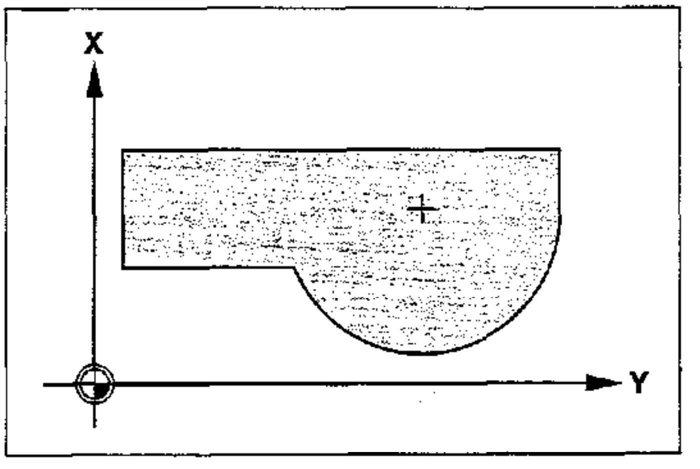


Fig. 5.1: A contour consists of straight lines and circular arcs

The contour elements are executed in sequence to machine the programmed contour.

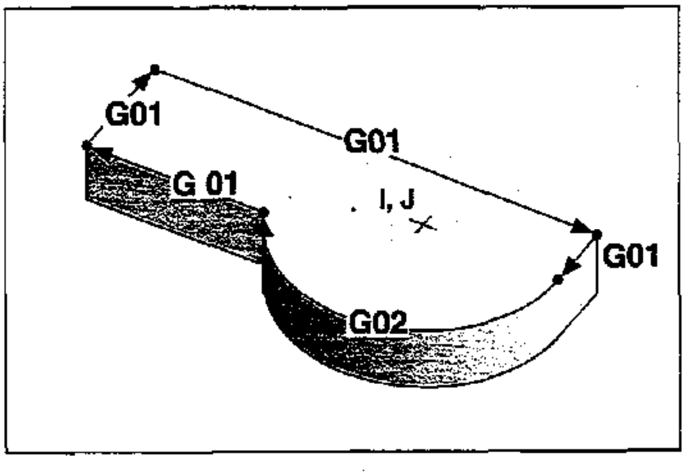


Fig. 5.2: Contour elements are programmed and executed in sequence

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 measurements
- 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.

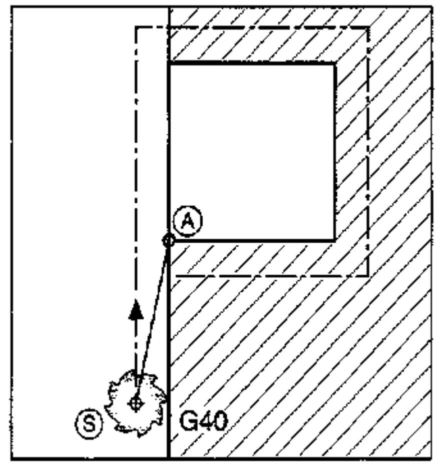


Fig. 5.3 : Starting point Ⓢ of machining

First contour point

Machining begins at the first contour point. The tool moves to this point with radius compensation.

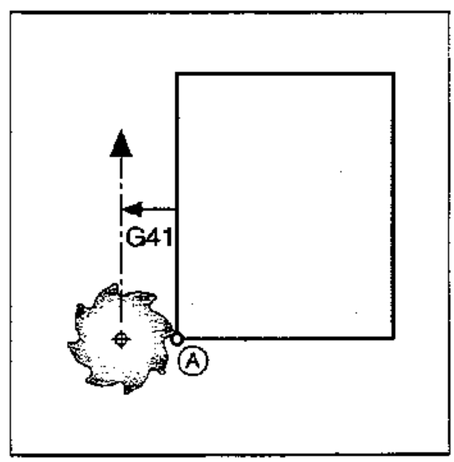


Fig. 5.4 : First contour point for machining

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

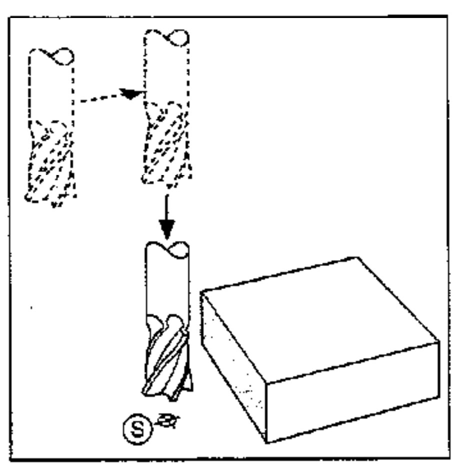


Fig. 5.5 : Separate movement of the spindle when there is danger of collision

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.

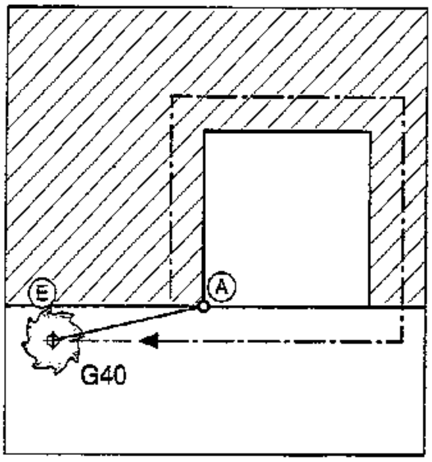


Fig. 5.6 : End point Ⓔ for machining

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

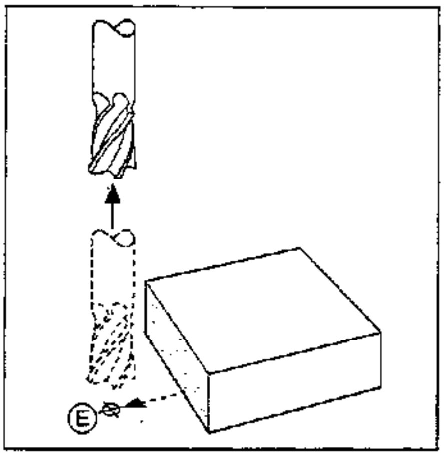


Fig. 5.7 : Retract spindle axis separately

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.

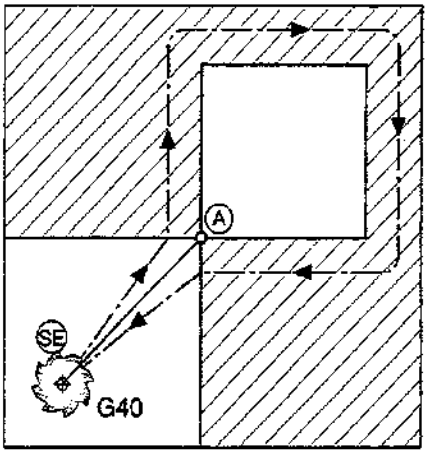


Fig. 5.8: .Common starting and end point

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).

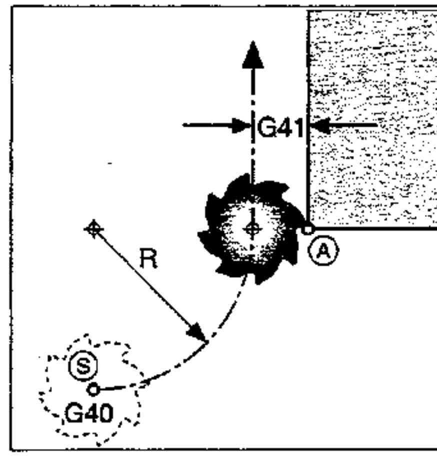


Fig. 5.9: Soft contour approach

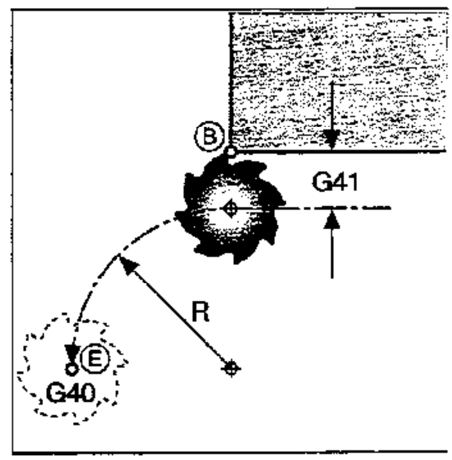


Fig. 5.10: Soft contour departure

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

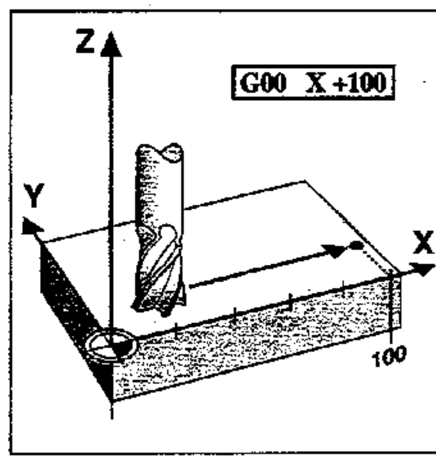


Fig. 5.11: Paraxial movement

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

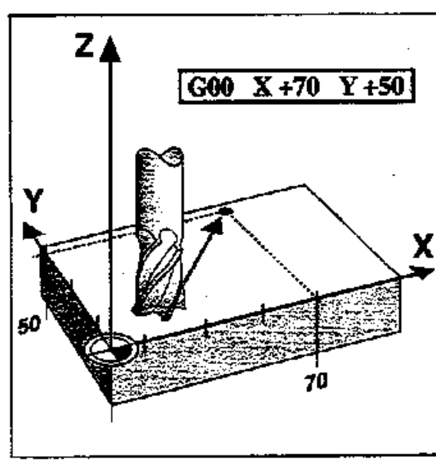


Fig. 5.12: Movement in a main plane (XY)

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.

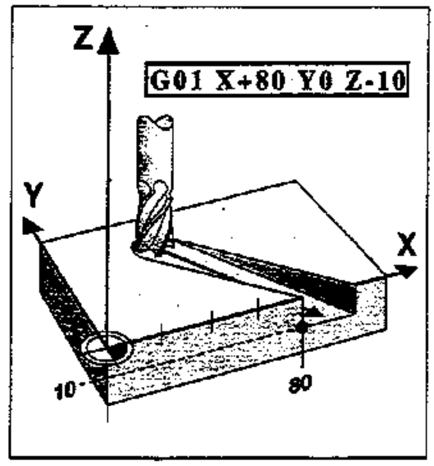


Fig. 5.13: Three-dimensional movement

Entering more than three coordinates (not TNC 407)

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

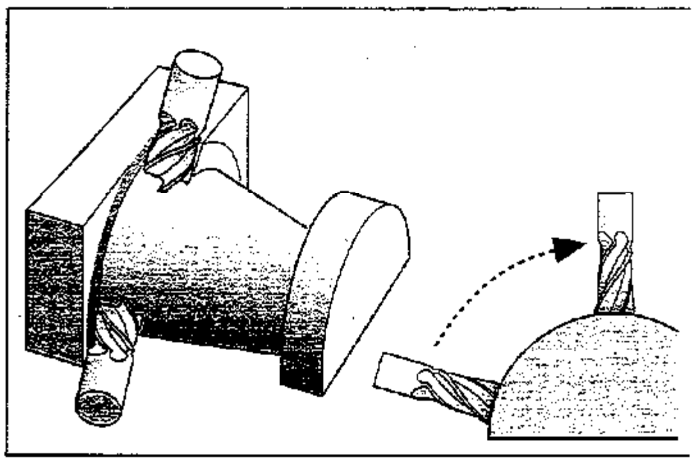


Fig. 5.14: Example of simultaneous movement of more than three axes: machining a 3D surface with an end mill

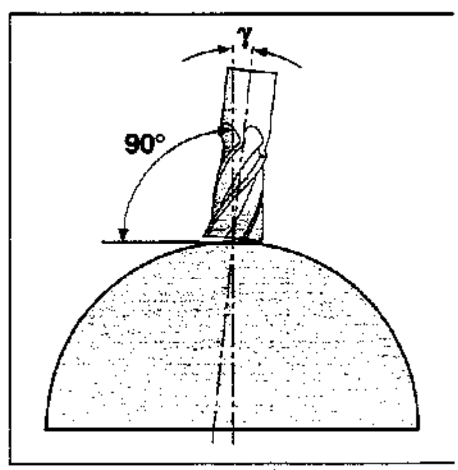


Fig. 5.15: Inclined-tool machining

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	G00	G10
Straight line at programmed feed rate	G01	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)	G02	G12
Circular arc, counterclockwise (CCW)	G03	G13
Programming of the circular path: • Circle center I, J, K and end point, or • Circle radius and end point.		
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.

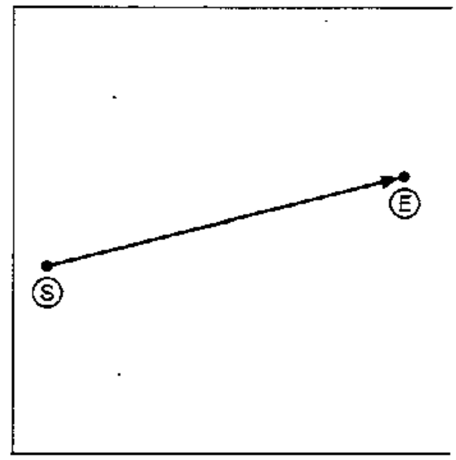


Fig. 5.16: Linear movement

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 coordinates of the end point
If necessary -/+	For negative coordinates, press the +/– key once, e.g. X = –50 mm
Y ⋮ Z	Enter all further coordinates of the end point

⋮

.
.
.

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.

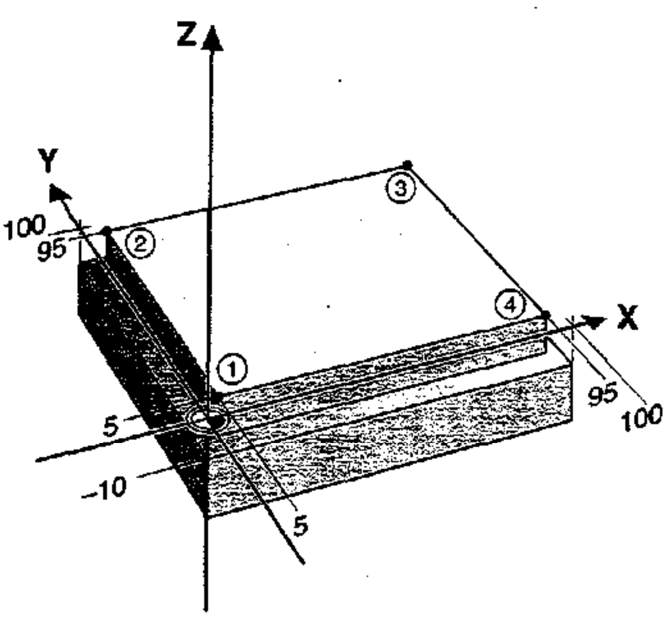
*Resulting NC block: N25 G00 G42 G91 X+50 G90 Y+10 Z-20 M3 **

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.

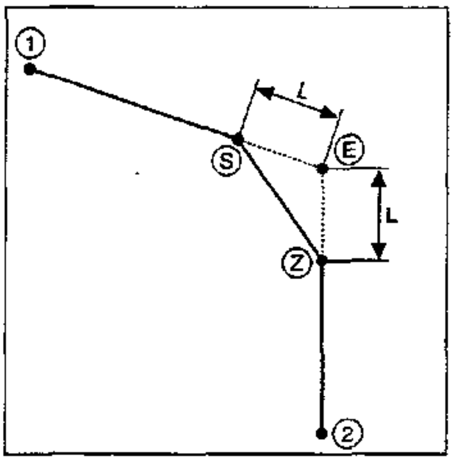


Fig. 5.17: Chamfer from Ⓢ to Ⓩ

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

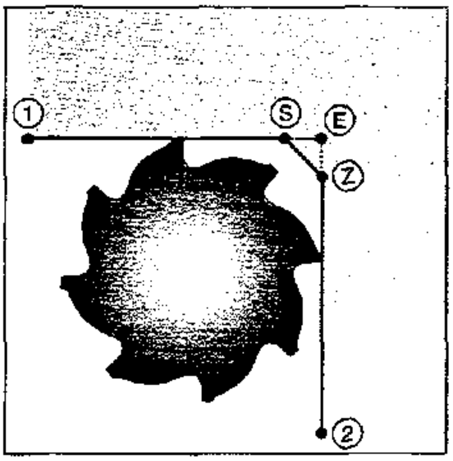


Fig. 5.18: Tool radius too large

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.

- 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.

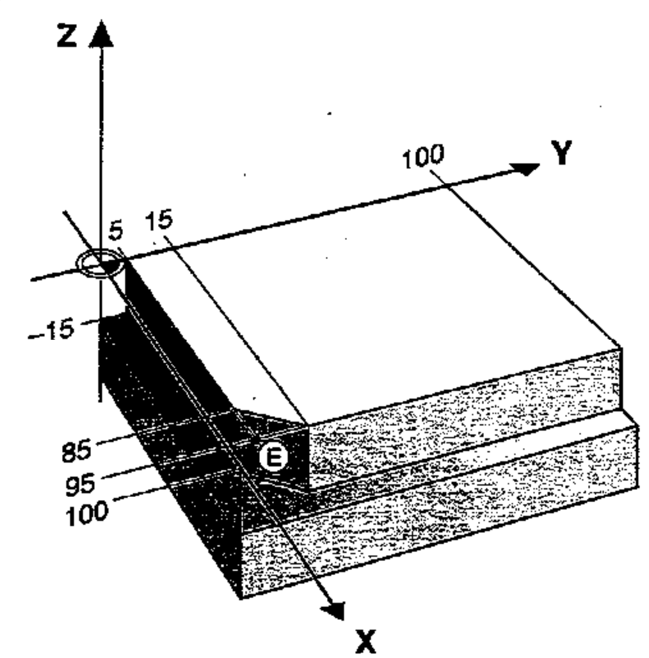
To program a chamfer:

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: Ⓔ	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 10mm
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.

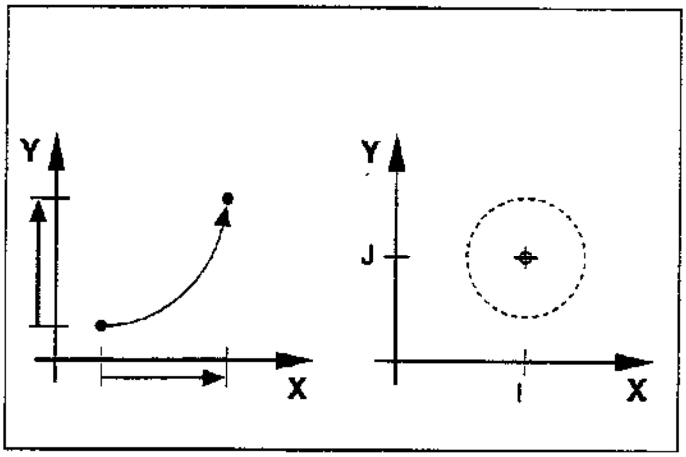


Fig. 5.19: Circular arc and circle center

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.

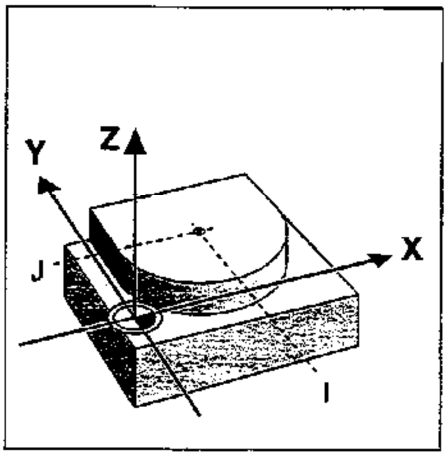


Fig. 5.20: Circle center coordinates

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

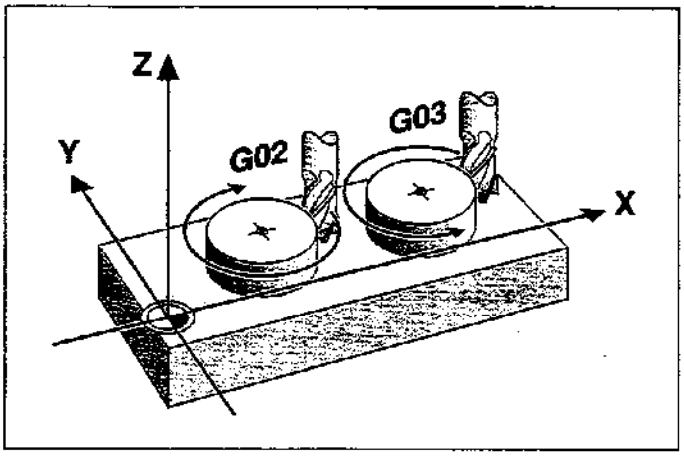


Fig. 5.21: Direction of rotation for circular movement

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	I J
Y	ZX G18	K I
X	YZ G19	J K

Fig. 5.22: Defining the spindle axis also defines the main plane

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.

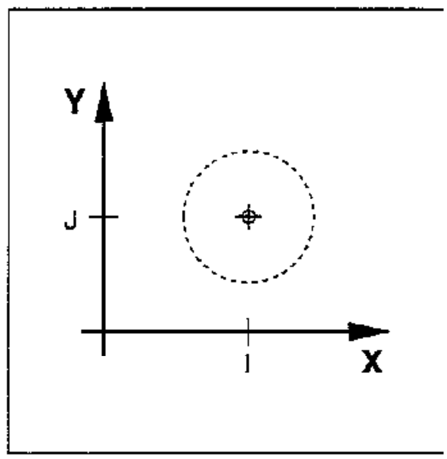


Fig. 5.23: Circle center I, J

Duration of circle center definition

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

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.

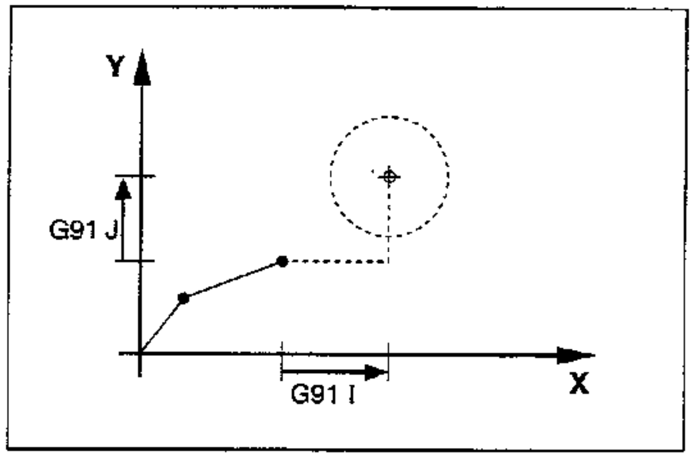


Fig. 5.24: Incremental coordinates for a circle center

- 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 **

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

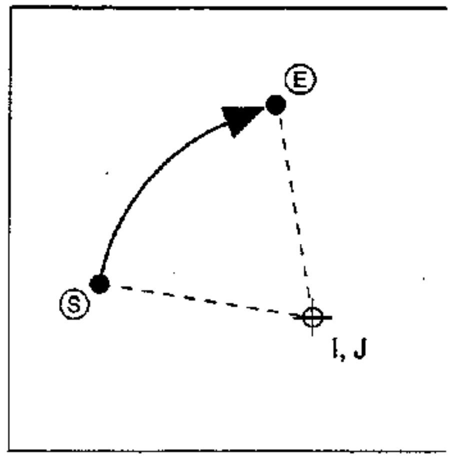


Fig. 5.25: Circular path from Ⓢ to Ⓔ around I, J

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

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.

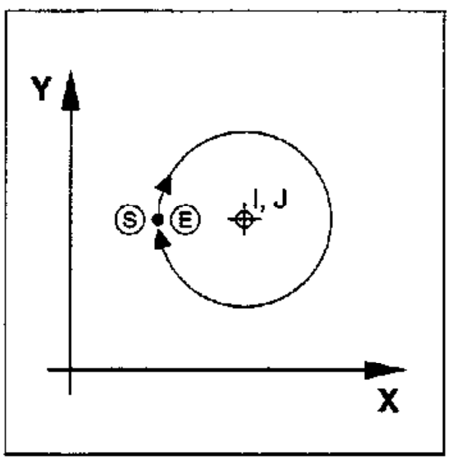


Fig. 5.26: Full circle around I, J with a G02 block

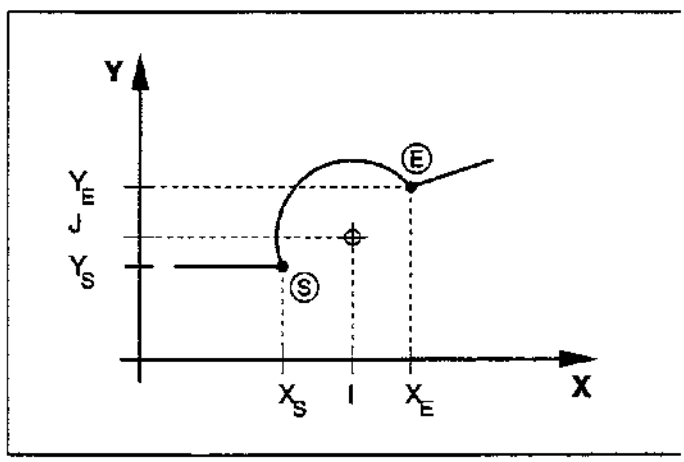


Fig. 5.27: Coordinates of an arc

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 -/+	Enter the second coordinate of the end point in absolute dimensions, for example, Y = –5 mm
END	Conclude the block

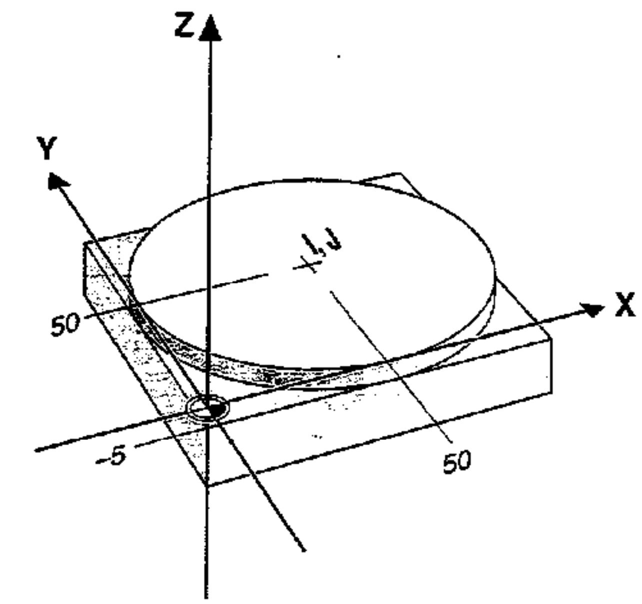
Further entries, if necessary:

- Radius compensation
- Feed rate
- Miscellaneous function

Resulting NC block: G02 G91 X+5 G90 Y–5

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 = +50mm, 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

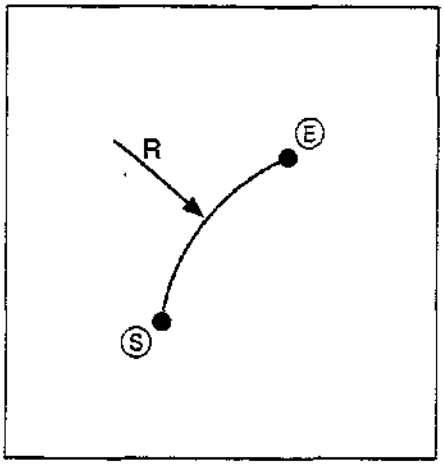


Fig. 5.28: Circular path from Ⓢ to Ⓔ with radius R

- 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.

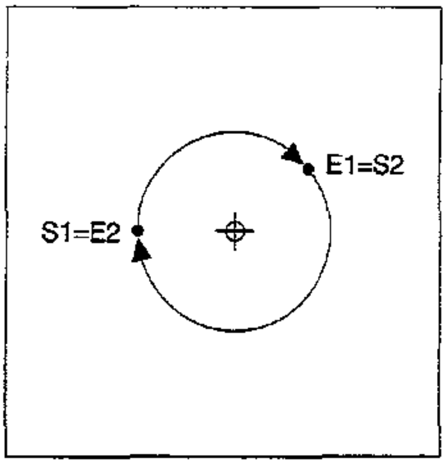


Fig. 5.29: Full circle with two G02 blocks

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).

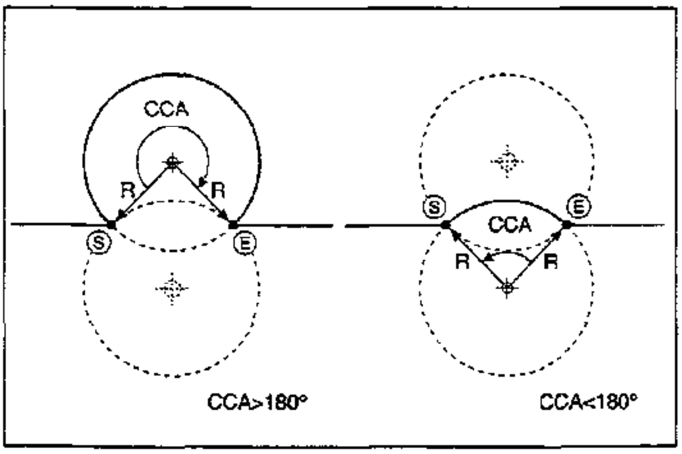


Fig. 5.30: Arcs with central angles greater than and less than 180°

Contour curvature and direction of rotation

The direction of rotation determines the type of arc:

- Convex (curving outward), or

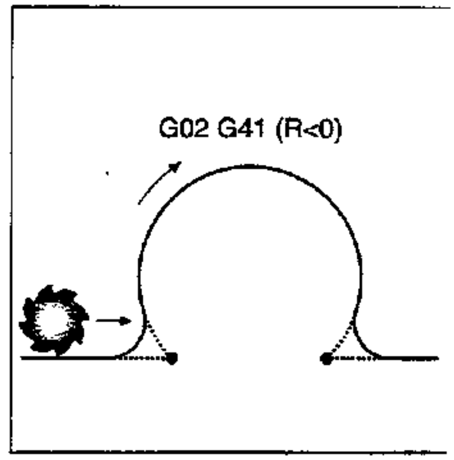


Fig. 5.31: Convex path

- Concave (curving inward)

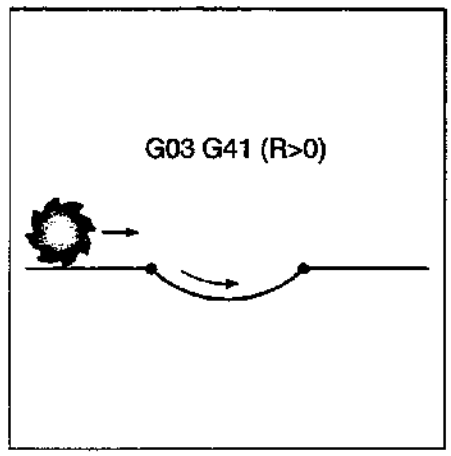


Fig. 5.32: Concave path

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)

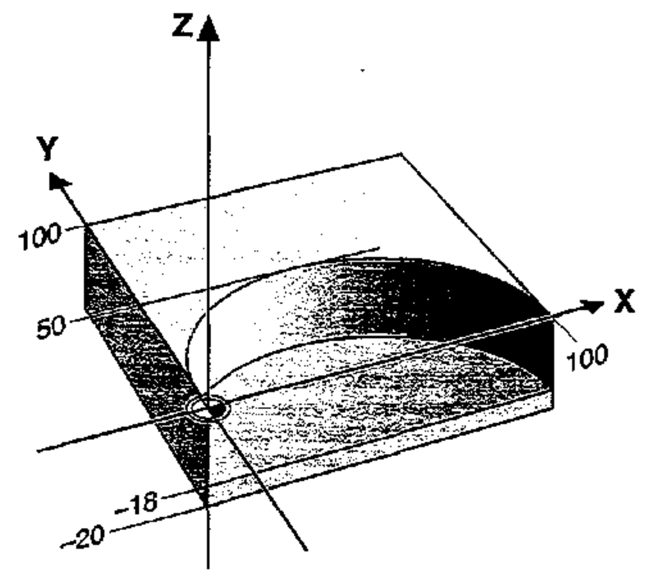
Further entries, if necessary:

- 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 = 0 Y = 0
Coordinates of the arc end point:	X = 100 mm Y = 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.

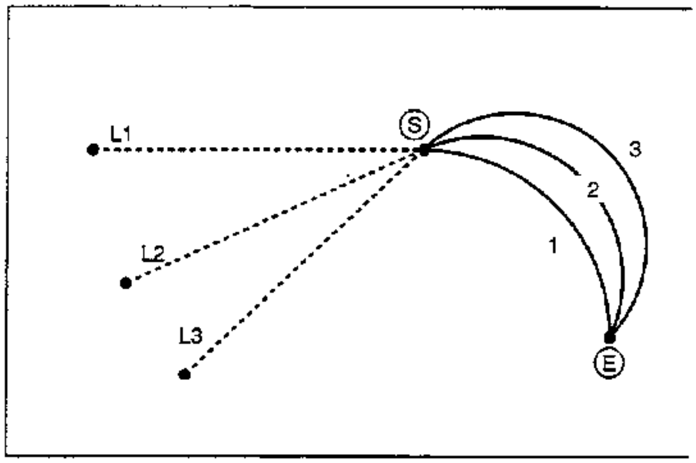


Fig. 5.33: The straight line ① - ② is connected tangentially to the circular arc Ⓢ - Ⓔ

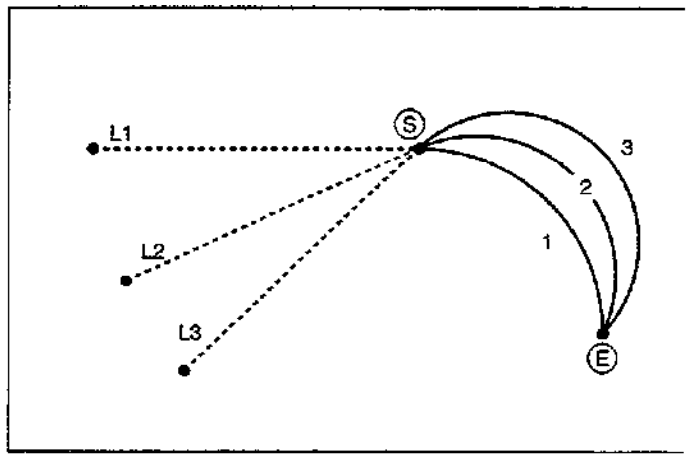


Fig. 5.34: The path of a tangential arc depends on the preceding contour element

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

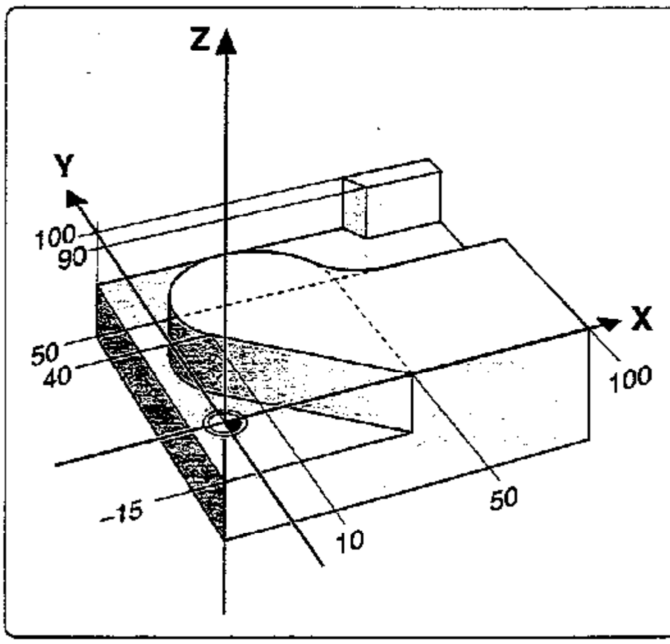
Further entries, if necessary:

- 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.

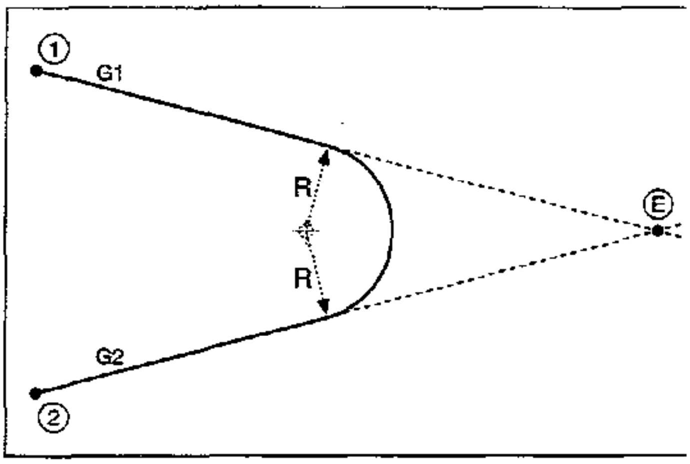


Fig. 5.35: Rounding radius R between G1 and G2

- 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.

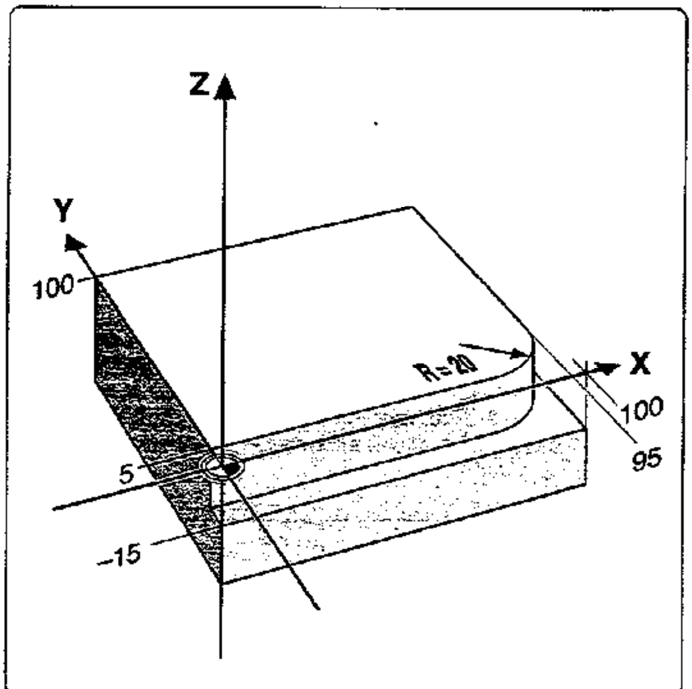
To program a tangential arc between two contour elements:

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 1-11).

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, I

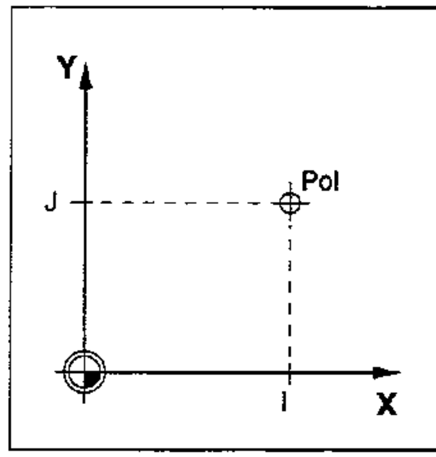


Fig. 5.36: The pole is the same as a circle center

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

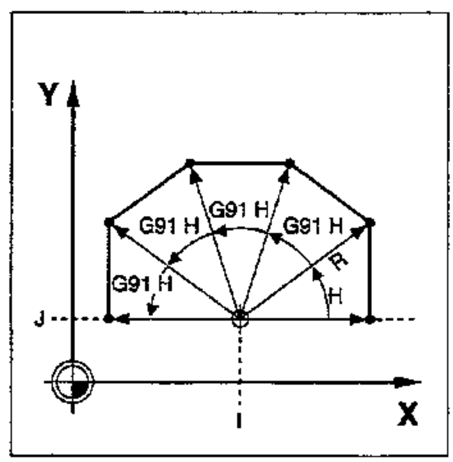


Fig. 5.37: Contour consisting of straight lines with polar coordinates

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 **

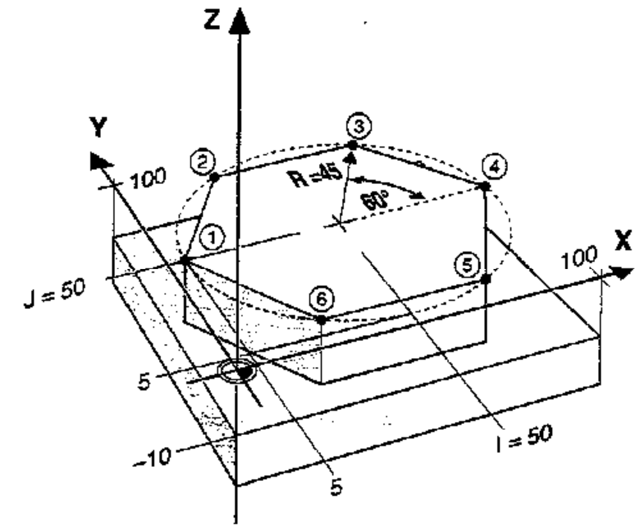
Practice exercise: Milling a hexagon

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

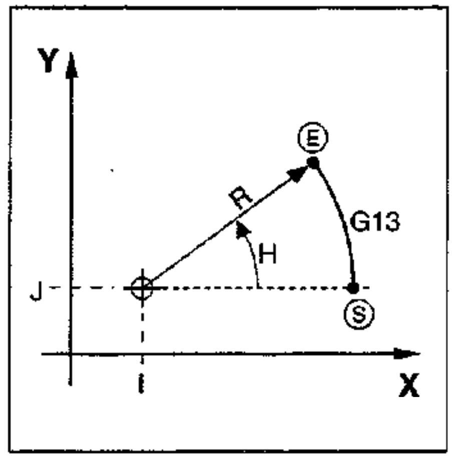


Fig. 5.38: Circular path around a pole

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°

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

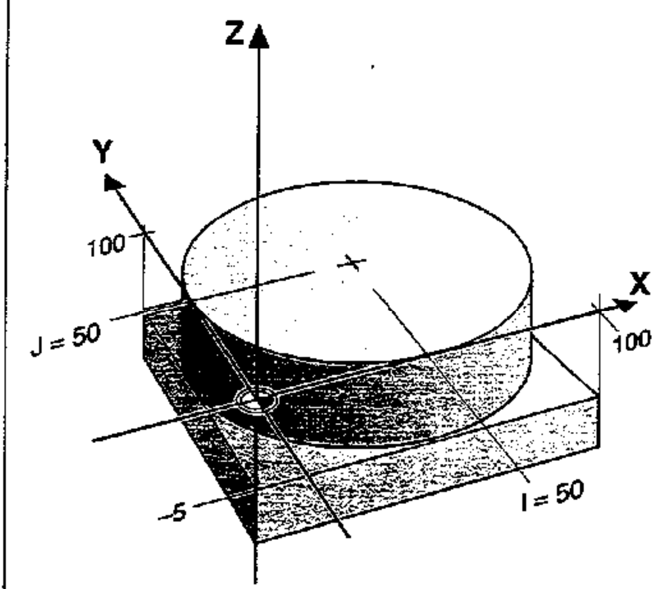
Further entries, if necessary:

Radius compensation R
Feed rate F
Miscellaneous function M

*Resulting NC block: G12 H30 **

Practice 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 Ⓔ

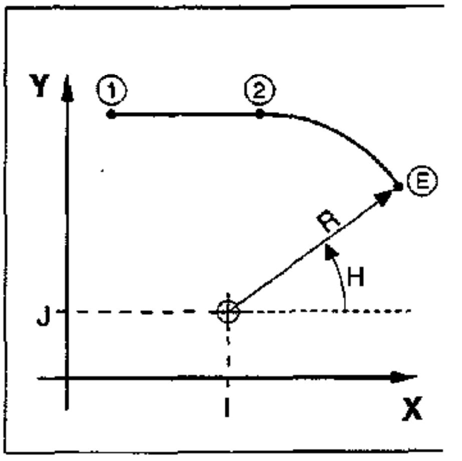


Fig. 5.39: Circular path around a pole with tangential transition

- 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

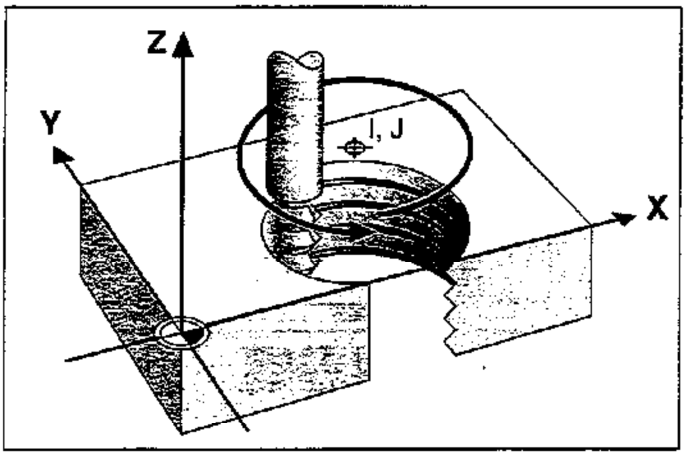


Fig. 5.40: A helix combines circular motion with linear motion

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

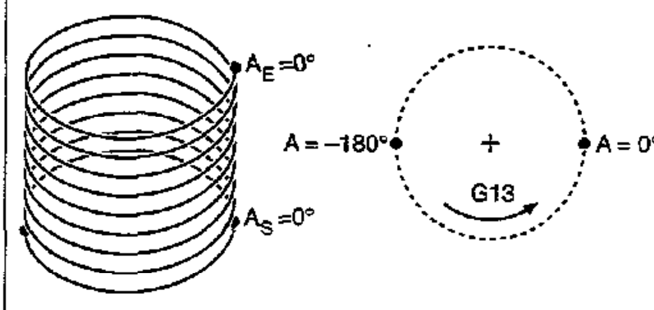
Thread:
Right-handed internal thread M64 x 1.5

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·n
  P = 1.5 mm
  n = $n_R + n_S + n_E$ = 9
  h = 13.5 mm

- Incremental polar coordinate angle H:

  H = n · 360°
  n = 9 (see total height h)
  H = 360°·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·($n_R + n_S$)
  = –1.5·8.5 mm
  = –12.75 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 *
```

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.

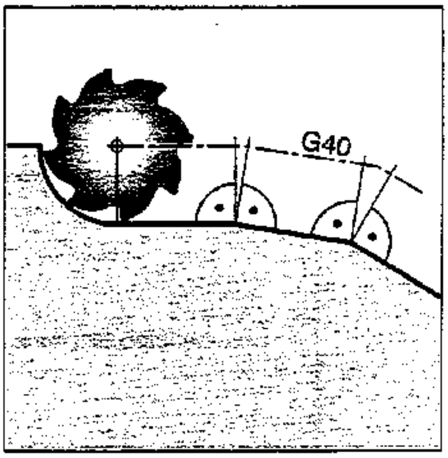


Fig. 5.42: Standard contouring behavior at G40 without M90

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.

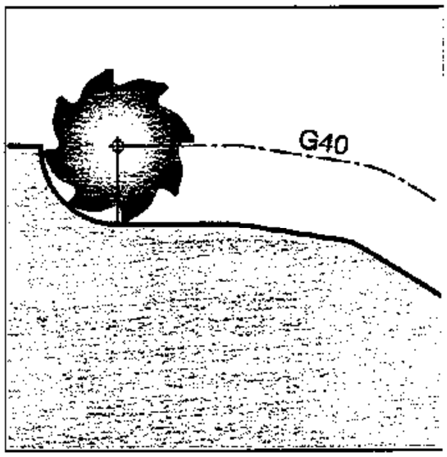


Fig. 5.43: Behavior at G40 with M90

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 11-13.

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.

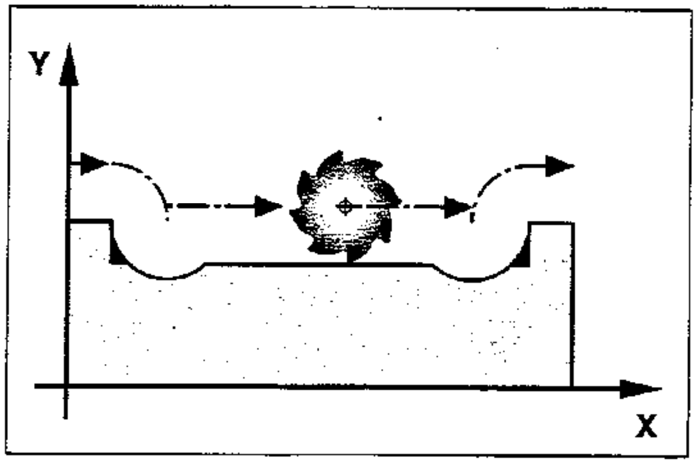


Fig. 5.44: Standard contouring behavior without M97 when the control would not generate an error message

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.

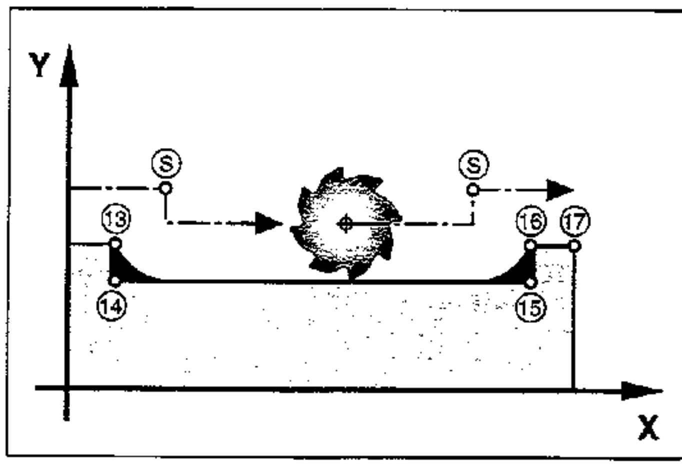


Fig. 5.45: Contouring behavior with M97

Duration of effect

M97 is effective only in the blocks in which it is programmed.

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 Ⓢ 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.

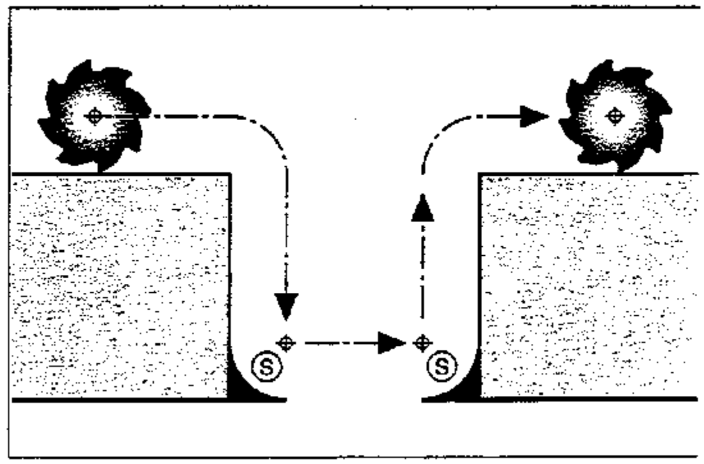


Fig. 5.46: Tool path without M98

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.

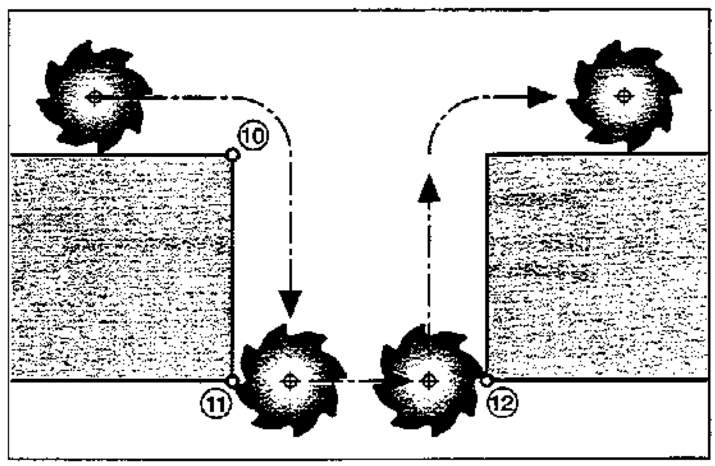


Fig. 5.47: Tool path with M98

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 1-12).

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.

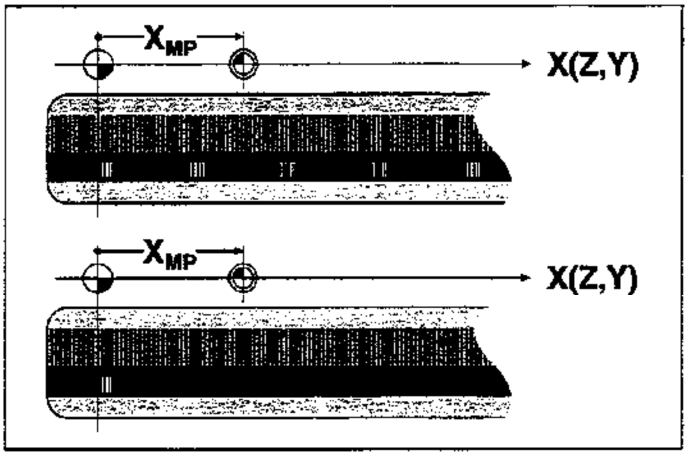


Fig. 5.48: Scale reference point and machine datum on scales with one or more reference marks.

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 2-6).

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.

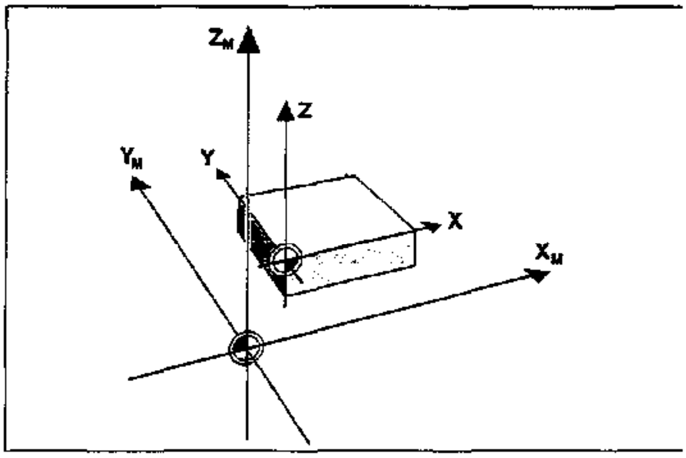


Fig. 5.49: Machine datum and workpiece datum

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

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.

Insert rounding arc between straight lines – with M112 E...

The TNC inserts a rounding arc between two straight lines. The size of the arc depends on the machine tool. It is calculated by the TNC such that the programmed feed rate (override setting 100%) is maintained at the rounded corner. If this is not possible, the TNC automatically decreases the feed rate.

You can enter a tolerance value E that defines the maximum permissible deviation from the programmed contour. When necessary, the TNC will reduce the feed rate in order to maintain the programmed tolerance.

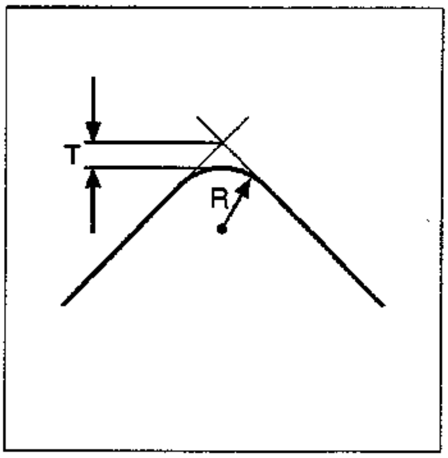


Fig. 5.50: Permissible deviation from the programmed contour

Duration of effect

M112 E.. is effective during operation with feed precontrol as well as with servo lag.

Cancelling

To cancel M112 E, enter M113.

Automatic compensation of machine geometry when working with tilted axes: M114 (not TNC 407)

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.

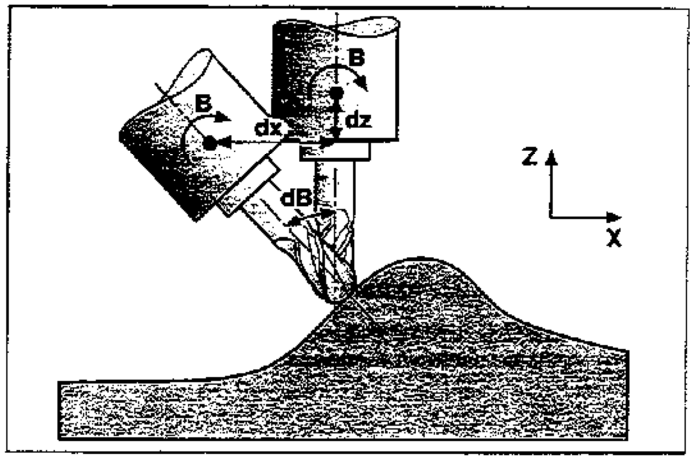


Fig. 5.51: Offset of the tool datum for tilting the tool

Automatic compensation of machine geometry – with M114

The TNC compensates the tool offset resulting from positioning with tilted axes. 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.

Cancelling

M114 is cancelled by M115 or by a N99 999 block.

The machine geometry must be defined by the machine builder 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 (END PGM block), whereupon it is automatically cancelled.

The machine geometry must be entered in machine parameters 7510 ff. by the machine tool builder.

Superimposing handwheel positioning during program run: M118 X... Y... Z...

Standard behavior – without M118

In the program run modes, the TNC moves the tool as defined in the part program.

Superimposing handwheel positioning with M118 X... Y... Z...

M118 enables manual adjustments to be made with the handwheel during program run. The range of this superimposed movement is entered behind M118 (in mm) in axis-specific values for X, Y and Z.

Cancelling

M118 X... Y... Z... is cancelled by entering M118 without the values for X, Y and Z.

Example

You wish to use the handwheel during program run to move the tool in the working plane X/Y by ±1 mm.

NC block: L X+0 Y+38.5 RL F125 M118 X1 Y1

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 MANUAL DATA INPUT
Select MDI operating mode

Program $MDI as desired

To execute the system file $MDI:

POSITIONING MANUAL DATA INPUT
Select POSITIONING MANUAL DATA INPUT operating mode

START	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 MANUAL 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
START	The rotary axis corrects the misalignment

6 Subprograms and Program Section Repeats

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

Sequence

The main program is executed up to the block in which a subprogram in 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).

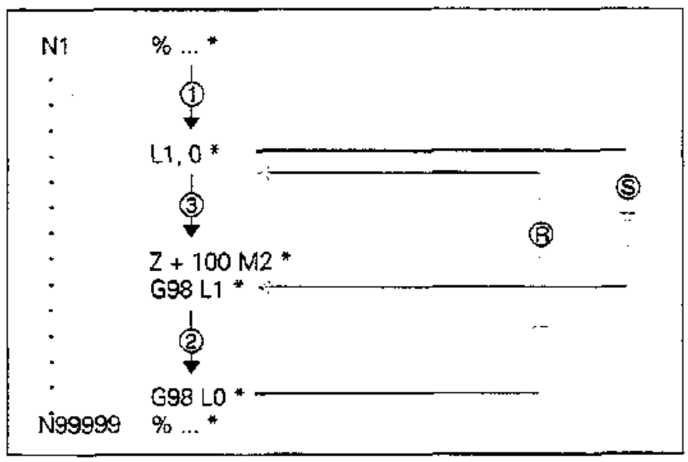


Fig. 6.1: Flow diagram for subprogramming
Ⓢ = jump Ⓡ = return jump

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 M2 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).

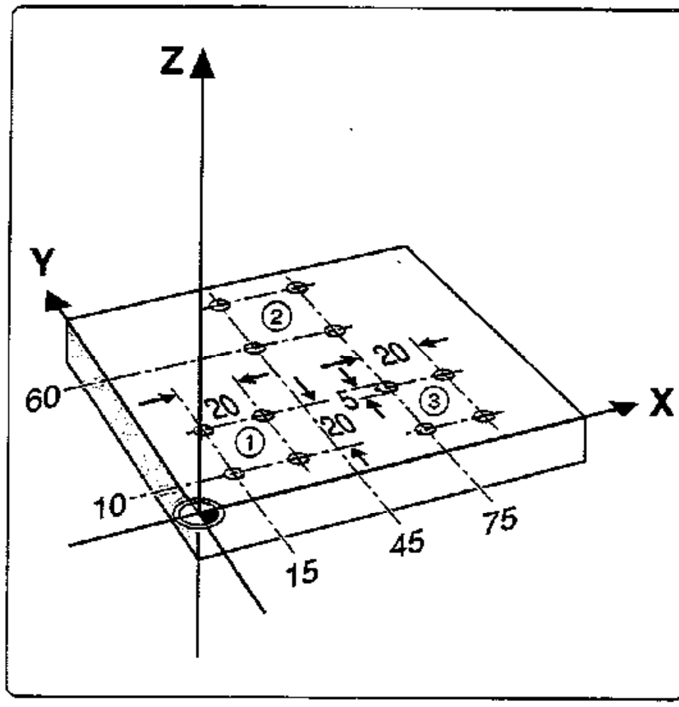
Coordinates of the first hole in each group:

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

%S641 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 (M2); the subprogram is entered behind M2
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 %S641 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).

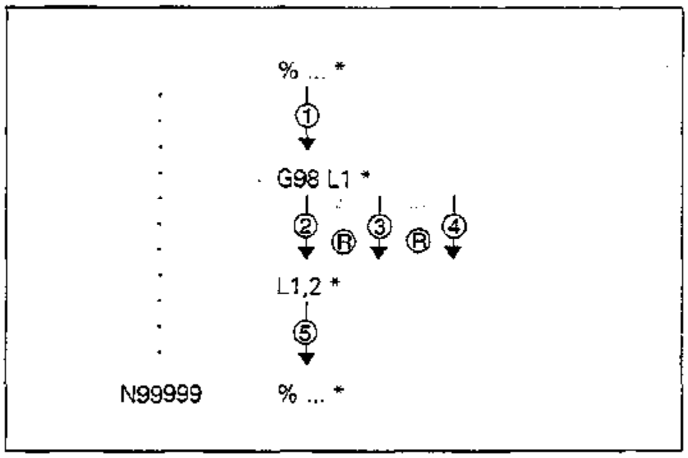


Fig. 6.2: Flow diagram for a program section repeat: Ⓡ = return jump

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.

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

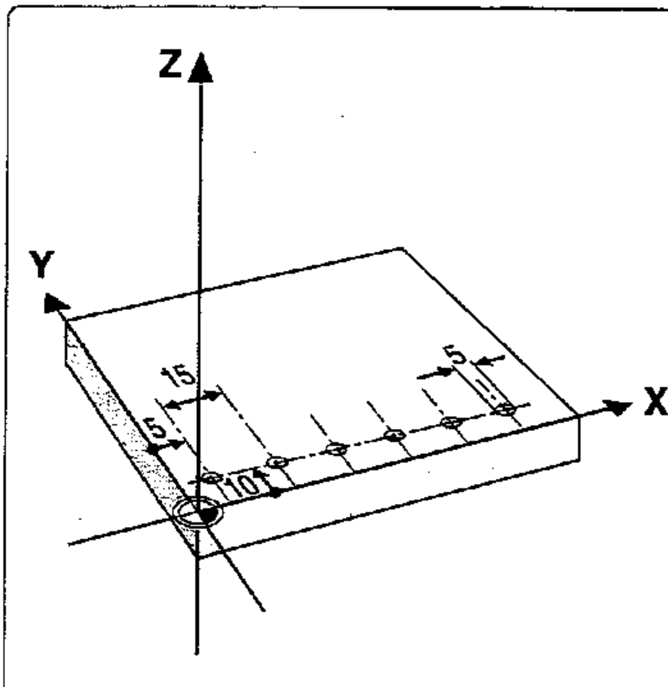
Enter the number of repeats in the block that calls the label. This is also the block that ends the program section.

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



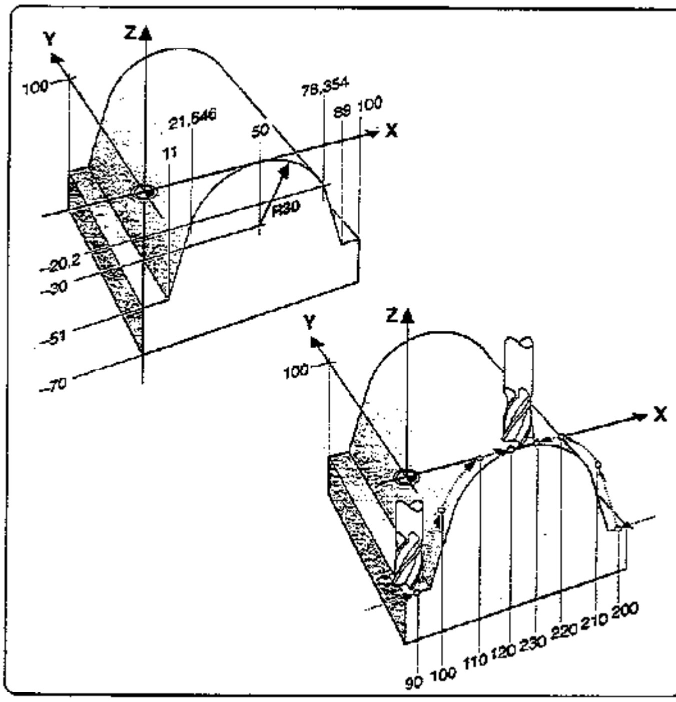
Part program

```
%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 Main Program as Subprogram

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).

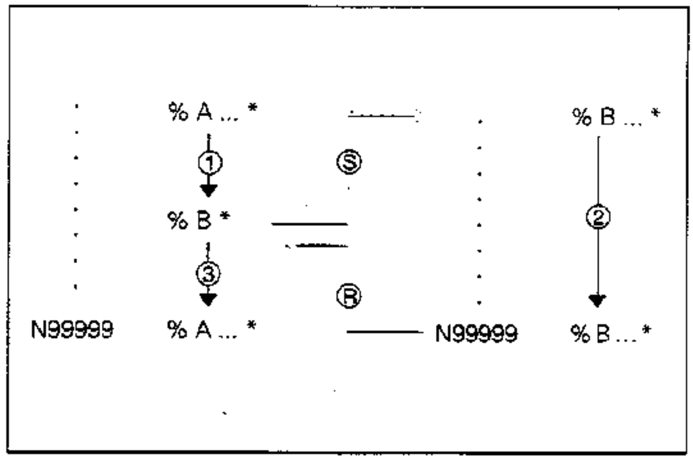


Fig. 6.3: Flow diagram of a main program as subprogram. Ⓢ = jump, Ⓡ = return jump

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 M2 or M30.
- The called program must not contain a jump into the calling program.

Calling a main program as a subprogram

% ▶ **PROGRAM NAME ?**

Enter the name of the program that you wish to call from this block.

EXT	.H	.I					

- 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 main program with cycle G39 (see page 8-48).

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
e.g. N39    L2,0 *                                    }  with call of
      .                                               }  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.

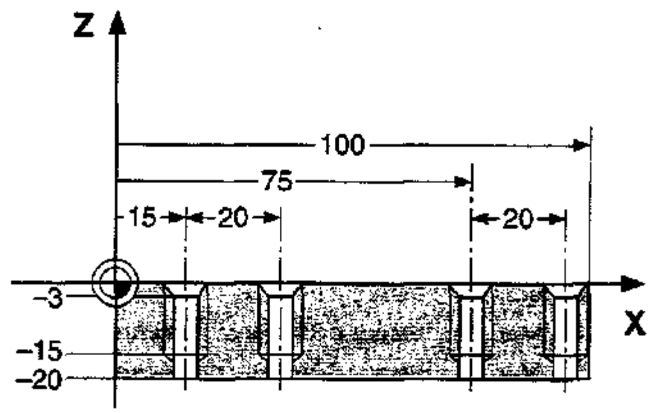
Example for exercise: Three groups of four holes (see page 6-4) with three different tools

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  1x4
e.g. N20  G98 L2 *
          1  2x2  1x4  2x5
e.g. N27  L2,2 * ............ The program section between this block and G98 L (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

7 Programming with Q Parameters

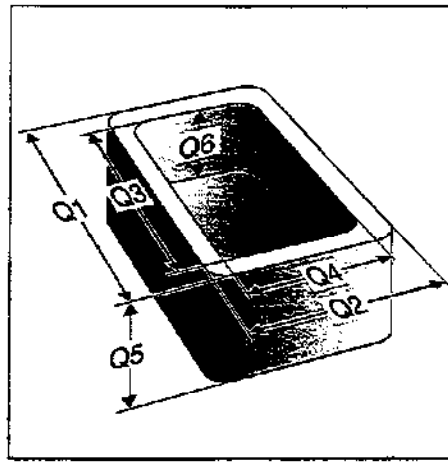


Fig. 7.1: Q parameters as variables

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 can represent information such as:

- coordinate values
- feed rates
- rpm
- cycle data

Q parameters are designated by the letter Q and a number between 0 and 119.

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.

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 12.

You can enter the individual Q parameter functions either blockwise (see page 7-7) or together in a formula through the ASCII keyboard (see page 7-16).

Use the soft key PARAMETER 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

Functions	**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 dimensions 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

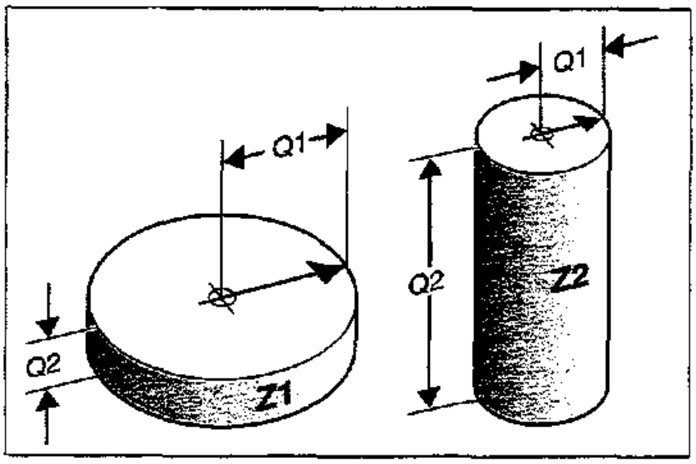


Fig. 7.2: Part dimensions as Q parameters

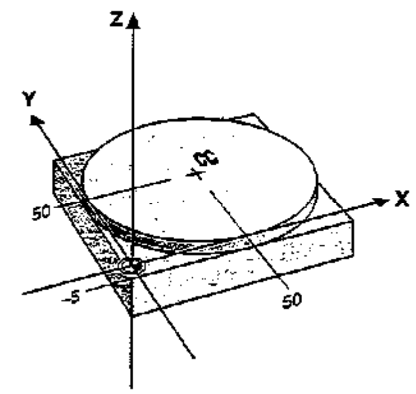
To assign numerical values to Q parameters:

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 *
```

Continued on next page...

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.

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.

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: *FN0: Q5 = +10*
*FN3: Q12 = +Q5 * +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:

$$\alpha = \arctan \alpha = \arctan (a/b) = \arctan (\sin \alpha / \cos \alpha)$$

Example: $a = 10$ mm
$b = 10$ mm
$\alpha = \arctan (a / b) = \arctan 1 = 45°$

Furthermore, $a^2 + b^2 = c^2 \quad (a^2 = a \cdot a)$
$c = \sqrt{a^2 + b^2}$

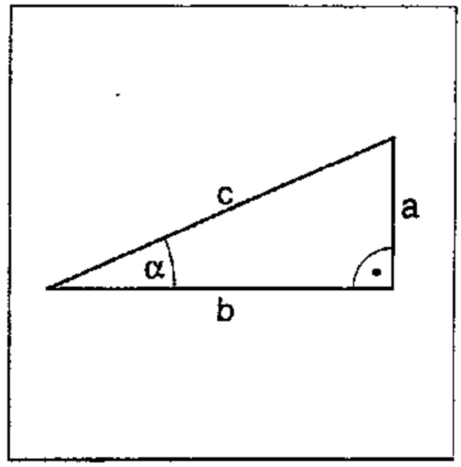


Fig. 7.3: Sides and angles on a right triangle

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° ≤ angle ≤ 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
.
.
.
15   G98 L5 * .................................... Label 5
16   % 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:

D14 ERROR=	D15 PRINT	D19 PLC=					END

Displaying error messages

D14 ERROR=

With the function D14: ERROR you can call messages that were pre-programmed 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 test 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

Your machine tool builder may have programmed a dialog text that differs from the above.

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. The TNC separates them with slashes.
Example: D15 P01 1 P02 Q1 P03 2 P04 Q2

Transfer to the PLC

D19 PLC=

The function D19: PLC transfers up to two numerical values or Q parameters 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/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

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 Q	ACOS
Arc tangent: Inverse of the tangent. Determine the angle from the ratio of the opposite to the adjacent side. Example: Q12 = ATAN Q11	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 in base 10 Example: Q33 = LOG Q22	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 x 3 + 2 x 10 = 35 =>	1st step: 5 x 3 = 15 2nd step: 2 x 10 = 20 3rd step: 15 + 20 = 35
Q2 = SQ 10 – 3^3 = 73 =>	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. Open 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.

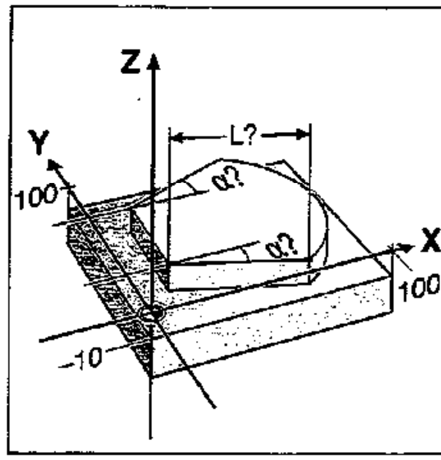


Fig. 7.4: Dimensions to be measured

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 11-12).

To program the use of a touch probe:

G 5 5 ENT

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?

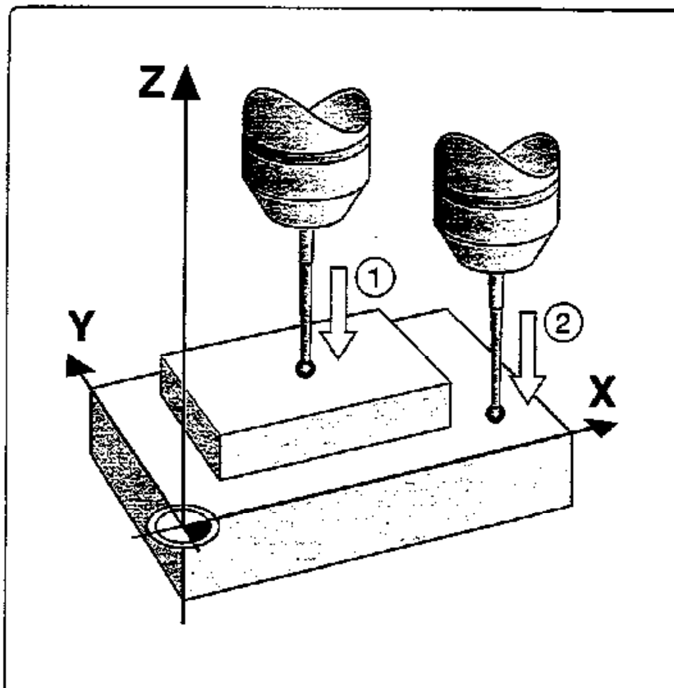
X	Enter the probing axis for the coordinate, for example X.
–/+ ENT	Select and confirm the probing direction.
X 5 Y 0 Z –/+ 5	Enter all coordinates for the pre-positioning point values, for example X = 5 mm, Y = 0, Z = –5 mm.
END	Conclude input.

*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)



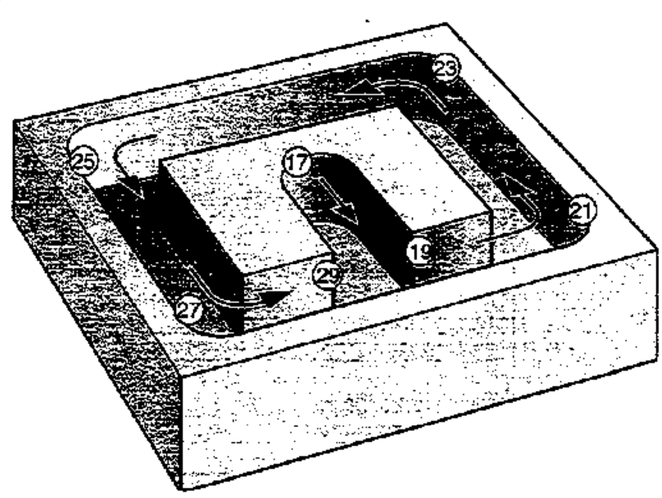
Part program

```
%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 7-14)
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
```

Continued on next page...

```
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 circle

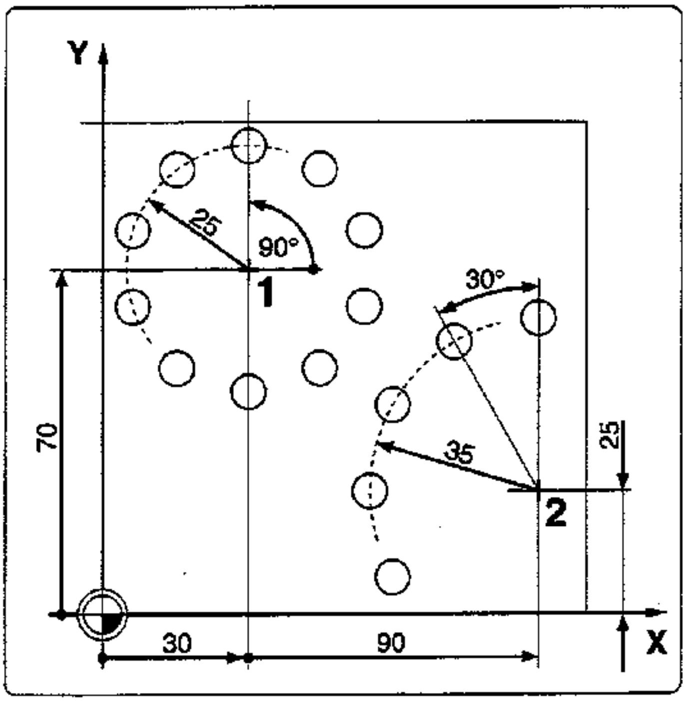
Bore pattern distributed over a full circle:

The entry values are listed in the program below in blocks N10–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– N190; Q5, Q7 and Q8 remain the same.



Part program

```
% LOCHKR G71 * ........................................ Load data for bolt hole circle 1
N10 D00 Q1 P01 +30 * .................................. Circle center X coordinate
N20 D00 Q2 P01 +70 * .................................. Circle center Y coordinate
N30 D00 Q3 P01 +11 * .................................. Number of holes
N40 D00 Q4 P01 +25 * .................................. Bolt circle radius
N50 D00 Q5 P01 +90 * .................................. Starting angle
N60 D00 Q6 P01 +0 * ................................... Hole angle increment (0: distribute holes over 360°)
N70 D00 Q7 P01 +2 * ................................... Setup clearance
N80 D00 Q8 P01 +15 * .................................. Total hole depth
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 * ....................................... Cycle definition: Pecking
N140 L1,0 * ........................................... Call bolt hole circle 1
                                                        Load data for bolt hole circle 2 (only re-enter changed
                                                        data)
N150 D00 Q1 P01 +90 * ................................. New circle center X coordinate
N160 D00 Q2 P01 +25 * ................................. New circle center Y coordinate
N170 D00 Q3 P01 5 * ................................... New number of holes
N180 D00 Q4 P01 +35* .................................. New bolt circle radius
N190 D00 Q6 P01 +30* .................................. New hole angle increment (not full circle, 5 holes 30° apart)
N200 L1,0 * ........................................... Call bolt hole circle 2
N210 G00 G40 G90 Z+200 M2 *
```

Continued on next page...

```
N220 G98 L1 * ........................................... Subprogram bolt hole circle
N230 D00 Q10 P01 +0 * ................................... Set the counter for finished holes
N240 D10 P01 +Q6 P07+QP03 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 +06 * .......................... 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 +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 Z+200 * ........................................ Retract in Z
N400 G98 L0 * ........................................... End of subprogram
N99999 % LOCHKR G71 *
```

Ellipse

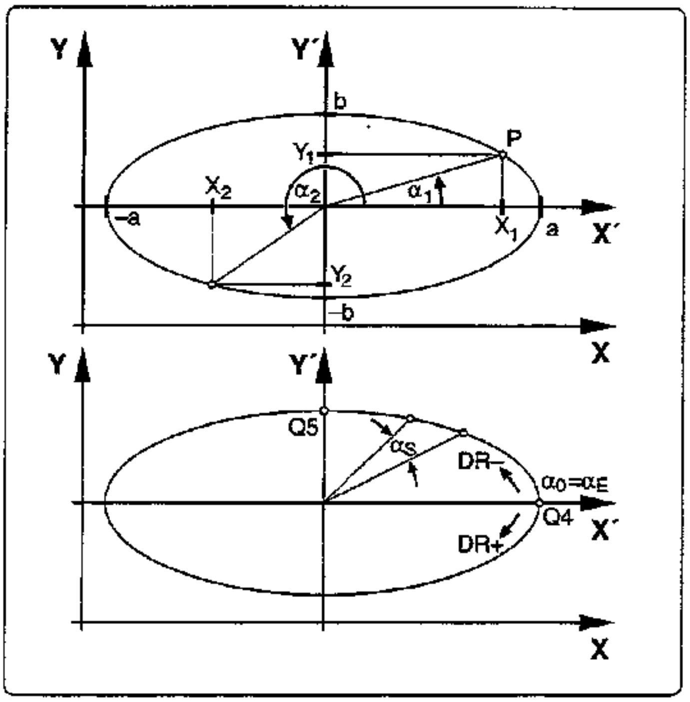
X-coordinate calculation: $X = a \cos \alpha$
Y-coordinate calculation: $Y = b \sin \alpha$

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_c, Y_c: 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...

```
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:

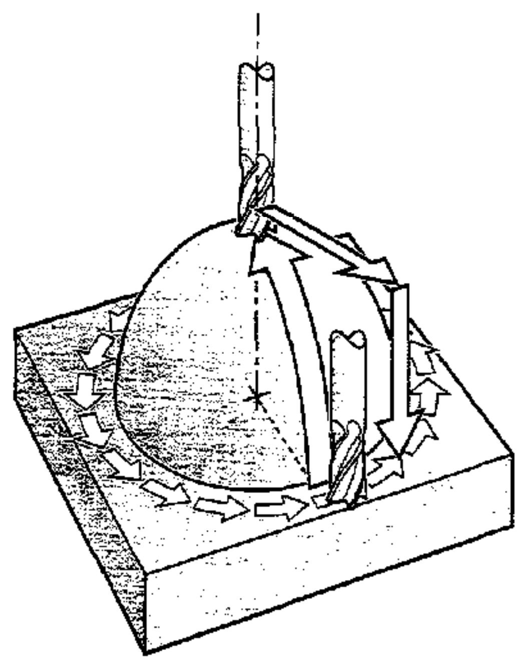
- The tool moves upward in the Z/X plane.
- You can enter an oversize in block 12 (Q12) if you want to machine the contour in several steps.
- The tool radius is automatically compensated with parameter Q108.

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 Cycles

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.
SET-UP 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 programmed 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
- 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)

Sequence:

- 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.

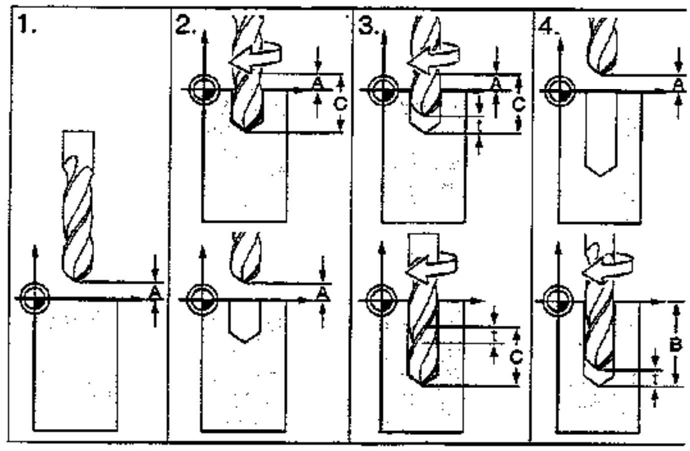


Fig. 8.1: PECKING cycle

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 F
  Traversing speed of the tool during drilling.

Calculations

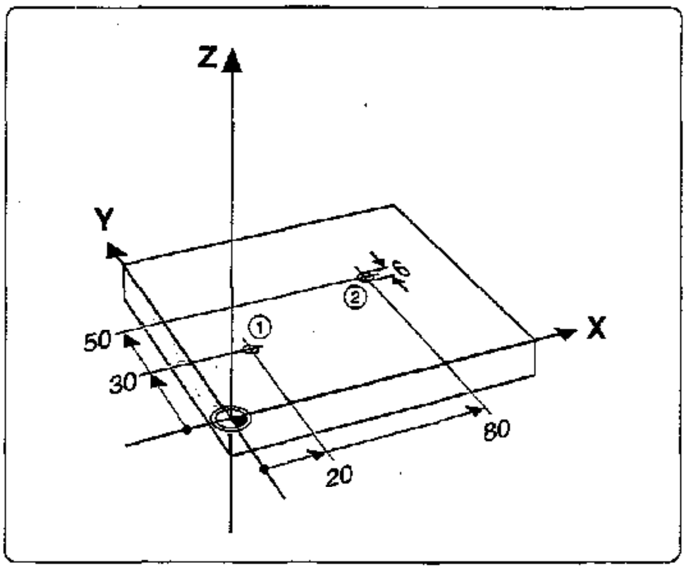
The advanced stop distance *t* is automatically calculated by the control:

- 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.

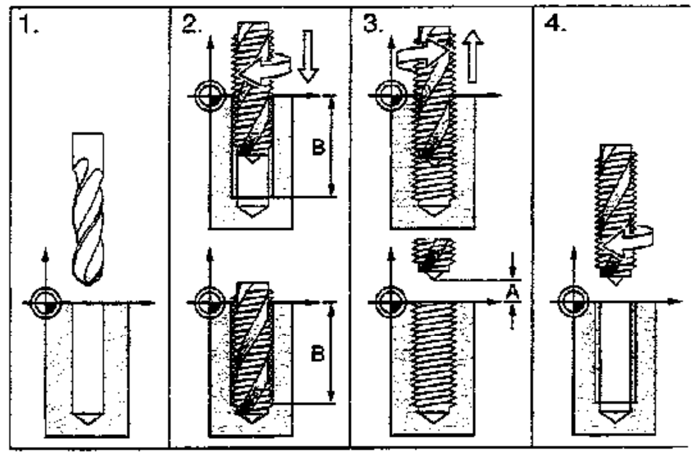


Fig. 8.2: TAPPING cycle

Required tool

A floating tap holder is required. It must compensate the tolerances between feed rate and spindle speed during the tapping process.

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 F:
  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

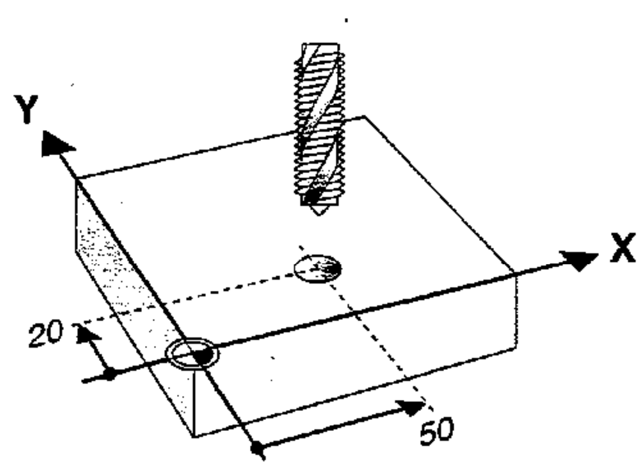
F = S x p ⇒ F = 100 · 1 = 100 mm/min

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)

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 11-12).
- Increased traverse range of the spindle axis due to absence of a floating tap holder

Machine and control must be specially prepared by the machine manufacturer to enable rigid tapping.

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

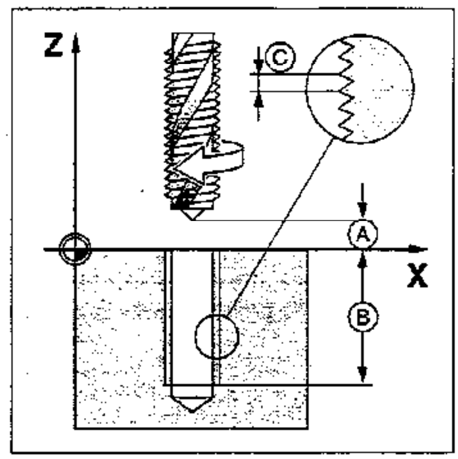


Fig. 8.3: Input data for RIGID TAPPING cycle

The control calculates the feed rate from the spindle speed and thread pitch. If the spindle speed override is used during tapping, the feed rate is automatically adjusted. The feed rate override knob is disabled.

THREAD CUTTING (G86)

Process

Thread cutting is performed by means of spindle control.
The spindle rotation is combined with linear movement in the tool axis, enabling helix-shaped cuts.

G86 THREAD CUTTING is adapted to the control and machine by the machine manufacturer, who can provide further information on this cycle.

Example

- Cutting an inner thread using a threading tool

The thread diameter depends on the tool used.

Input data

- DEPTH: Distance between workpiece surface and end of thread
- PITCH: Thread pitch

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: (slit 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.

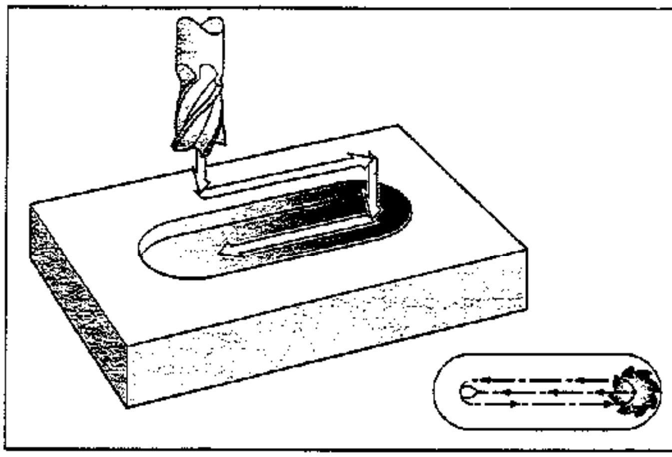


Fig. 8.4: SLOT MILLING cycle

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.

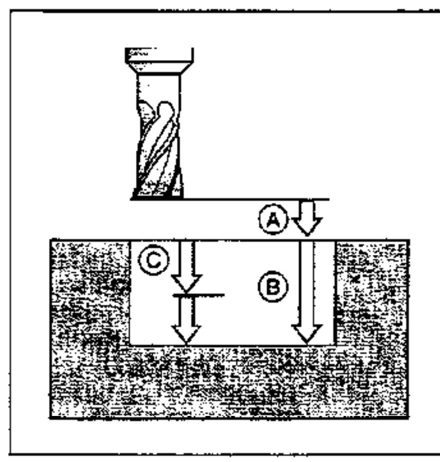


Fig. 8.5: Infeeds and distances for the SLOT MILLING cycle

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.

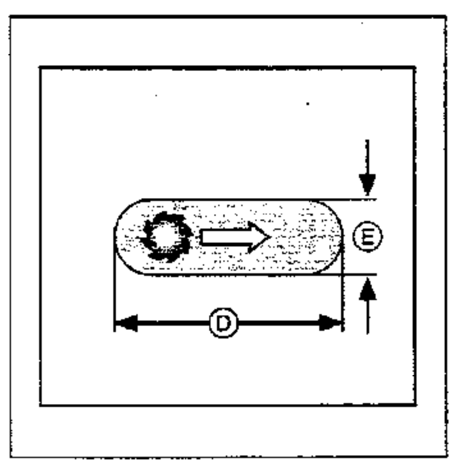


Fig. 8.6: Side lengths of the slot

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.

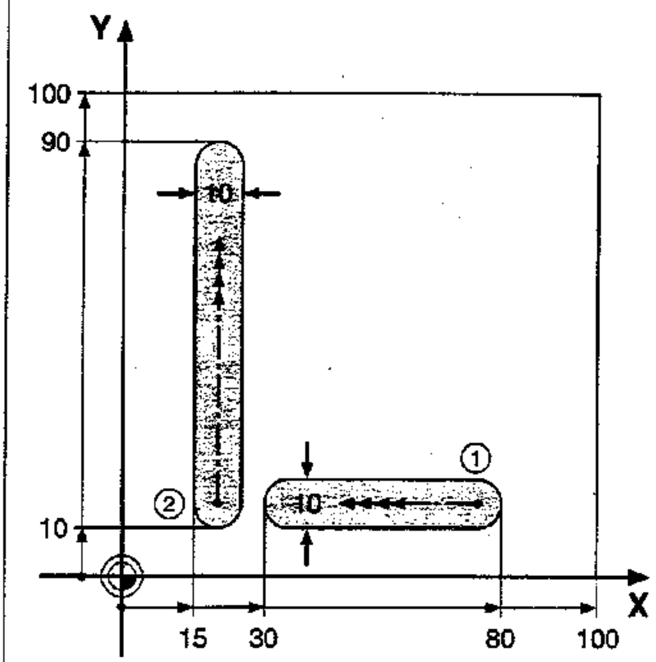
Starting position, slot ①:
X = 76 mm Y = 15 mm

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 8-9)

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.

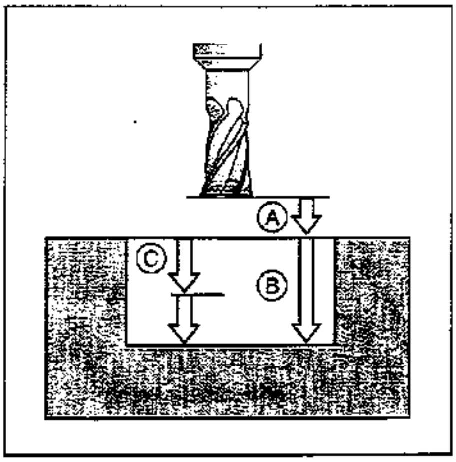


Fig. 8.7: Infeeds and distances for the POCKET MILLING cycle

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

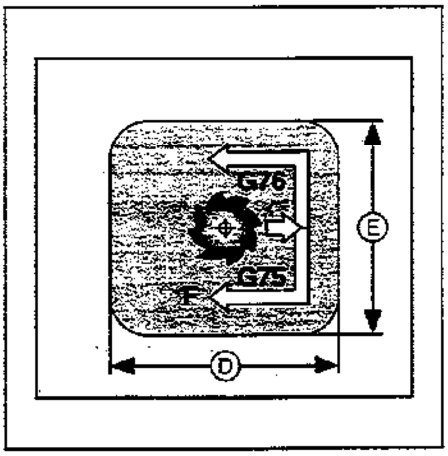


Fig. 8.8: Side lengths of the pocket

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.

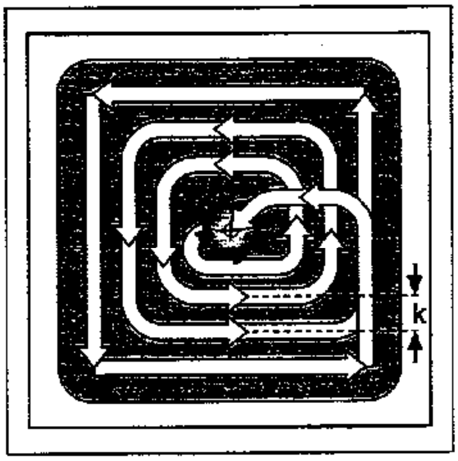


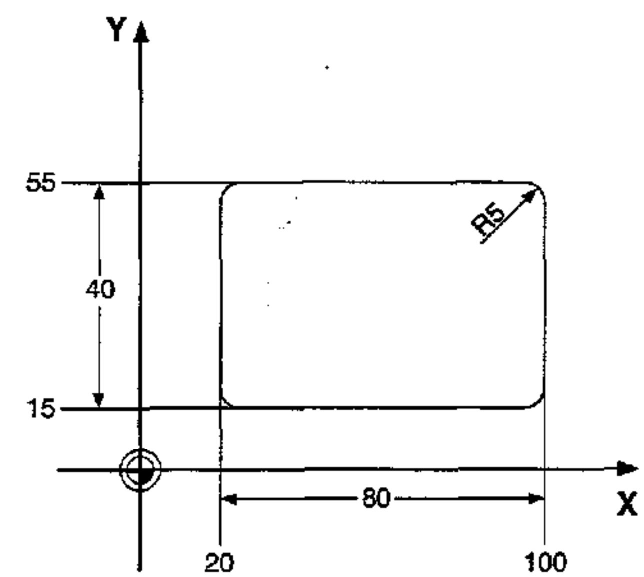
Fig. 8.9: Tool path for roughing-out

Corner radius

The corner radius is determined by the radius of the milling tool.

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 8.10) 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

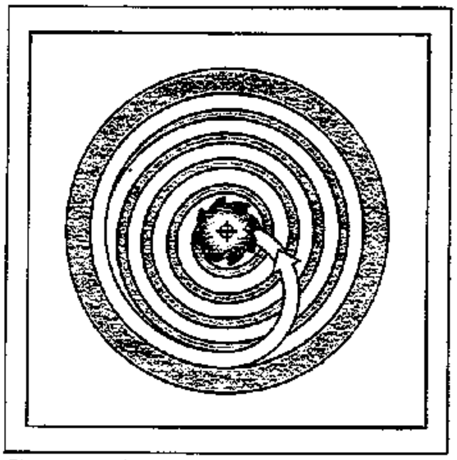


Fig. 8.10: Cutter path for roughing-out

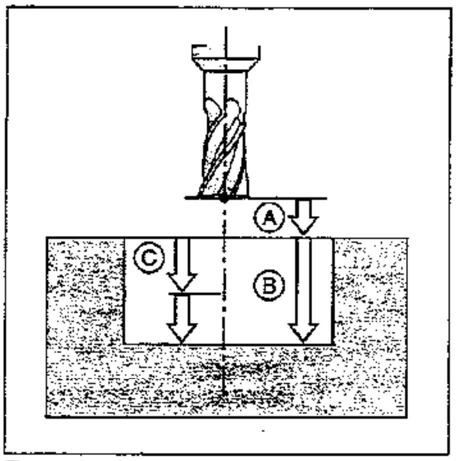


Fig. 8.11: Distances and infeeds for CIRCULAR POCKET MILLING

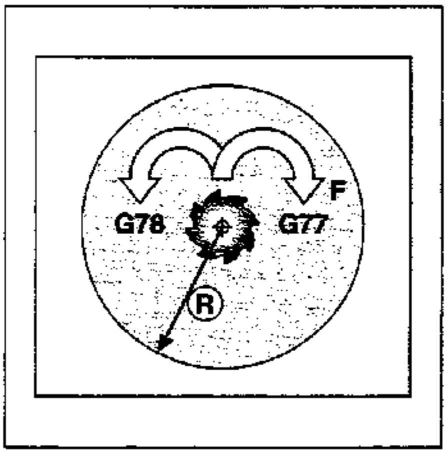


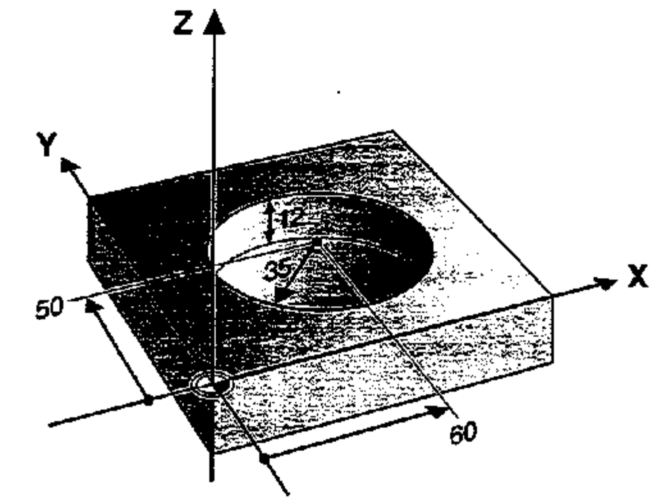
Fig. 8.12: Direction of the cutter path

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.
- 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)

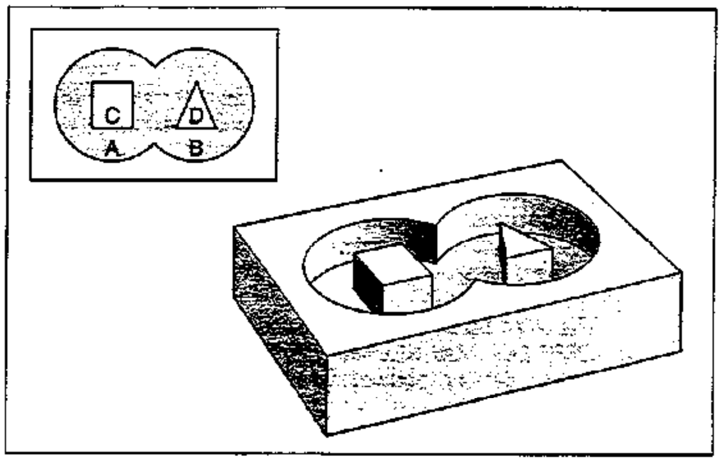


Fig. 8.13: Example of an SL contour. *A* and *B* are pockets, *C* and *D* are islands

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.

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.

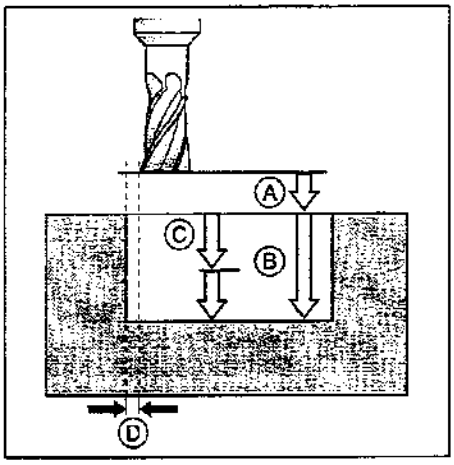


Fig. 8.14: Infeeds and distances of the ROUGH-OUT cycle

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

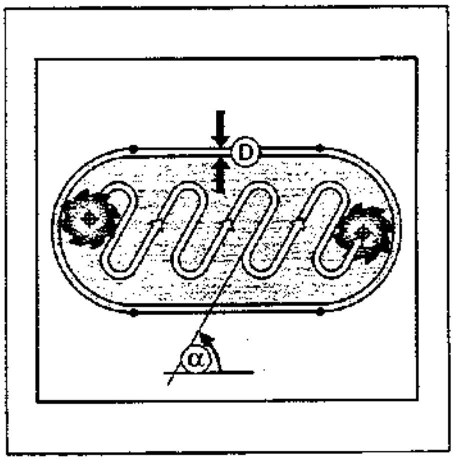


Fig. 8.15: Cutter path for roughing-out

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

Example: Roughing-out a rectangular pocket

Rectangular pocket 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

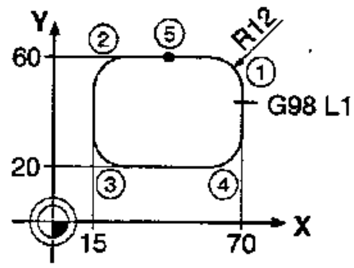
Coordinates of the auxiliary pocket:

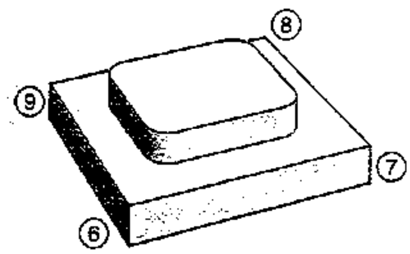
	X	Y
⑥	–5 mm	–5 mm
⑦	105 mm	–5 mm
⑧	105 mm	105 mm
⑨	–5 mm	105 mm

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 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.

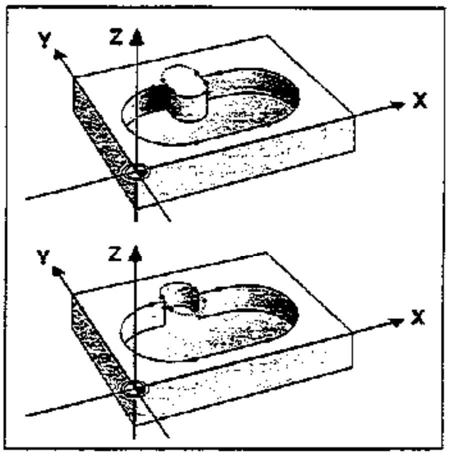


Fig. 8.16: Examples of overlapping contours

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

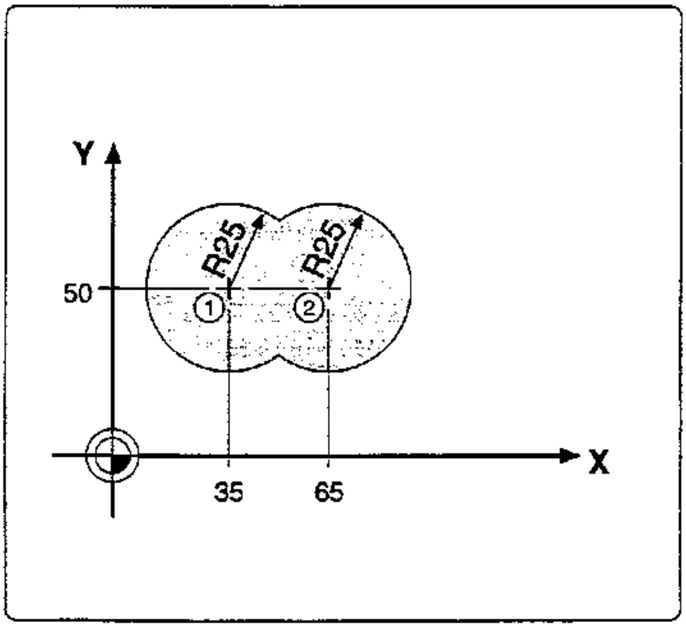
Coordinates of the circle centers:

① X = 35 mm Y = 50 mm
② X = 65 mm Y = 50 mm

Circle radii

R = 25 mm

Safety 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



Continued on next page...

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 *
```

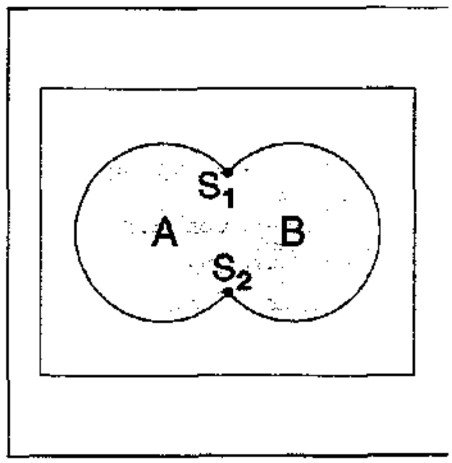


Fig. 8.17: Points of intersection S_1 and S_2 of pockets *A* and *B*

Depending on the control setup (machine parameters), machining starts either with the outline or the surface:

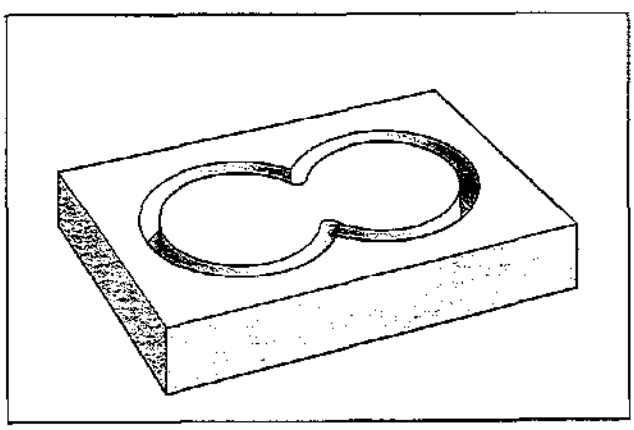


Fig. 8.18: Outline is machined first

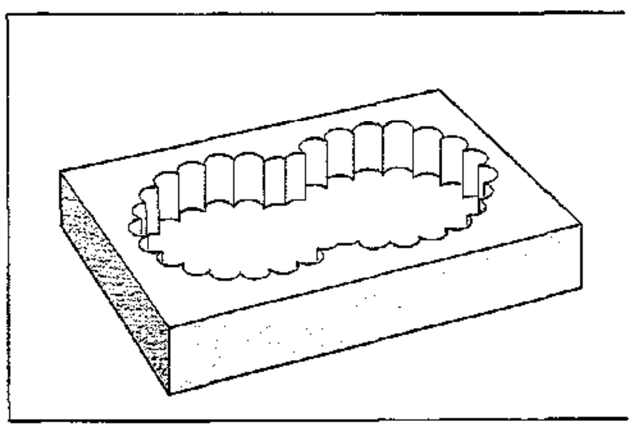


Fig. 8.19: Surface is machined first

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 *
```

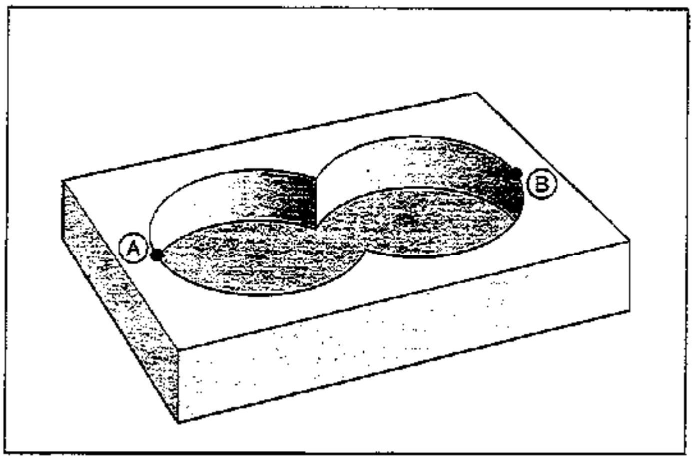


Fig. 8.20: Overlapping pockets: area of inclusion

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 *
```

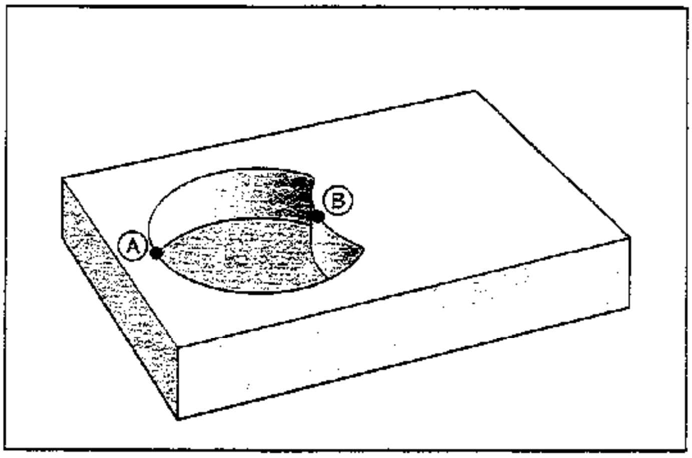


Fig. 8.21: Overlapping pockets: area of exclusion

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 *
```

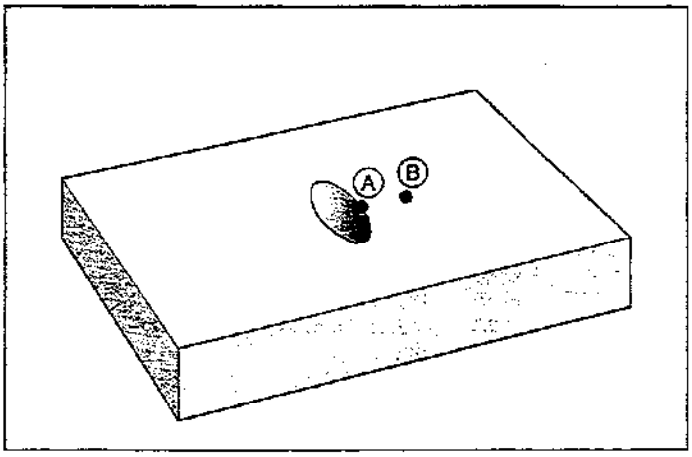


Fig. 8.22: Overlapping pockets: area of intersection

The subprograms are used in the main program on page 8-20.

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
```

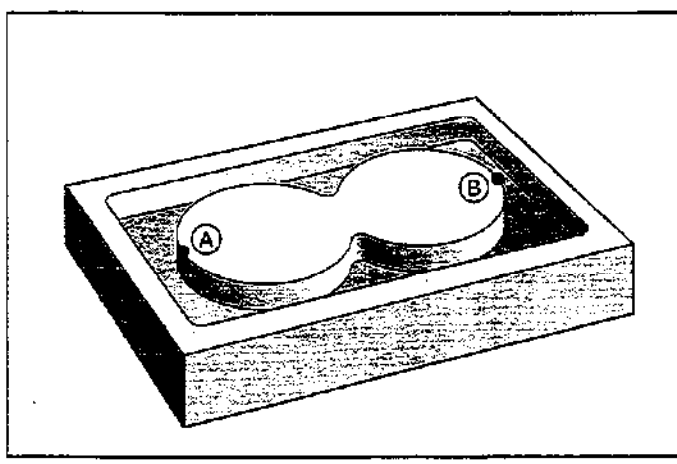


Fig. 8.23: Overlapping islands: area of inclusion

The subprograms and supplements are entered in the main program on page 8-22.

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 lie 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*
```

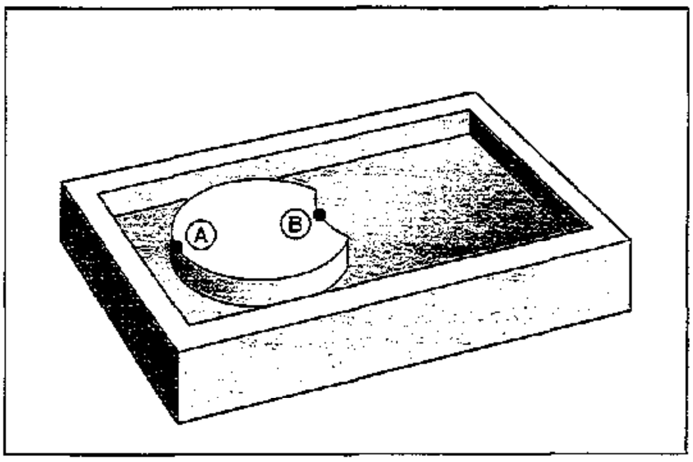


Fig. 8.24: Overlapping islands: area of exclusion

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
```

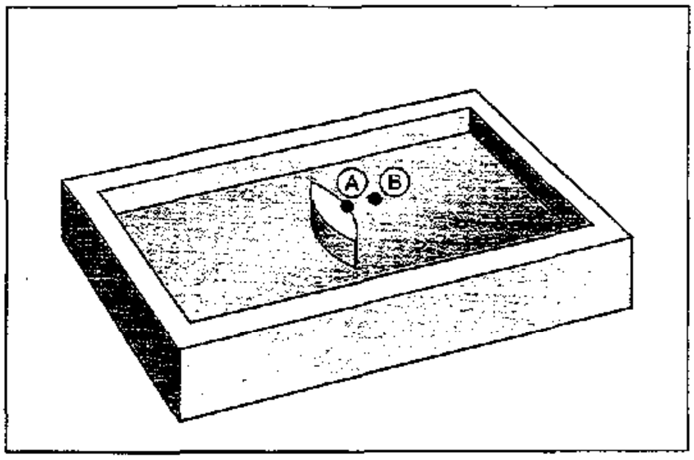


Fig. 8.25: Overlapping islands: area of intersection

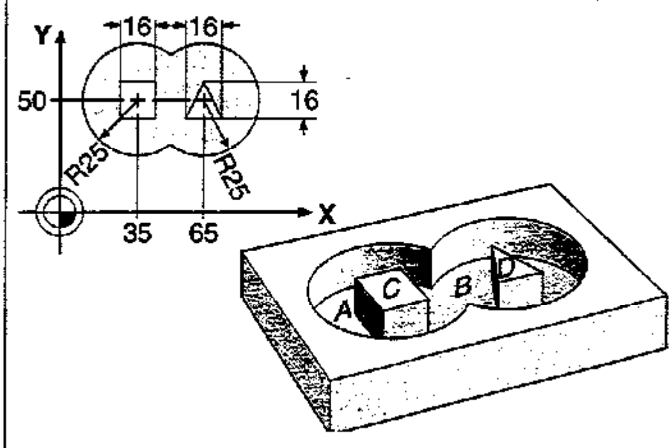
Example: Overlapping pockets and islands

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 *
```

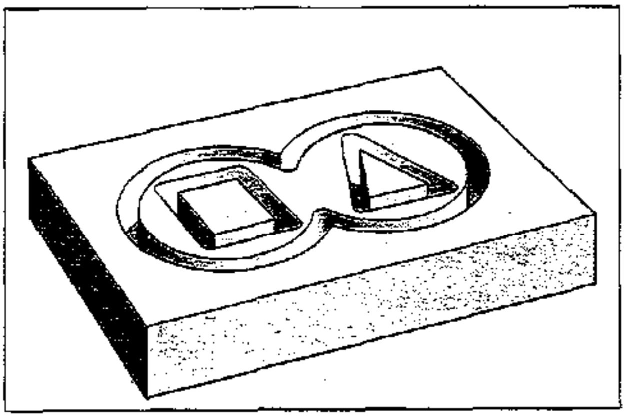


Fig. 8.26: Milling of outline

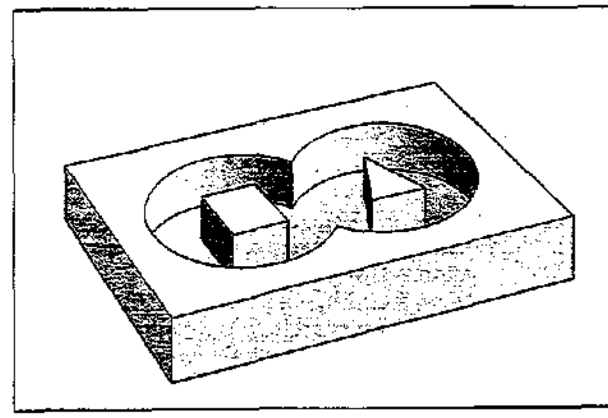


Fig. 8.27: Finished workpiece

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.

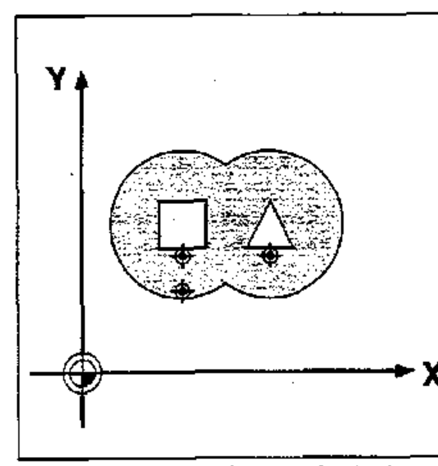


Fig. 8.28: Example of cutter infeed points for PECKING

Input data

- SETUP CLEARANCE
- TOTAL HOLE DEPTH
- PECKING DEPTH
- DWELL TIME
- FEED RATE
- FINISHING ALLOWANCE Ⓓ

$\left.\right\}$ identical to cycle G83 PECKING

Allowed material for the drilling operation (see figure 8.29).
The sum of the tool radius and the finishing allowance should be the same for pilot drilling as for roughing out.

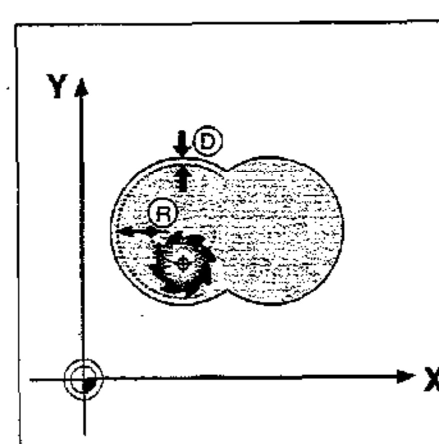


Fig. 8.29: Finishing allowance

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.

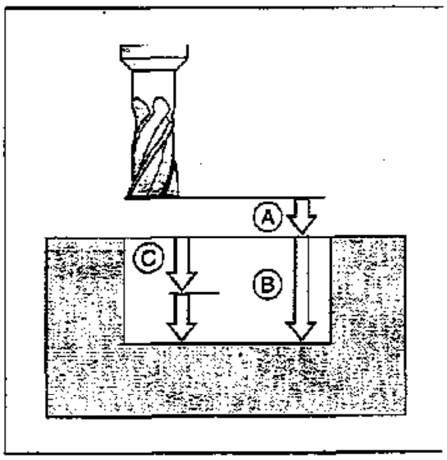


Fig. 8.30: Infeeds and distances for CONTOUR MILLING

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

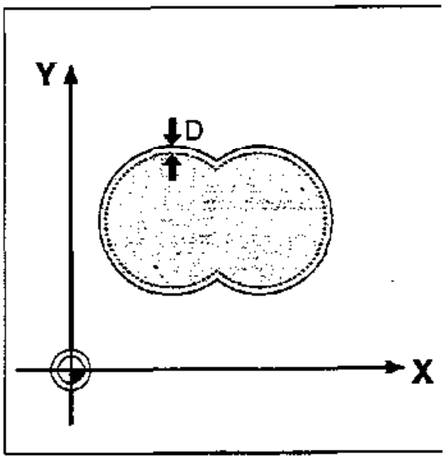


Fig. 8.31: Finishing allowance

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

The following scheme illustrates the application of thé 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

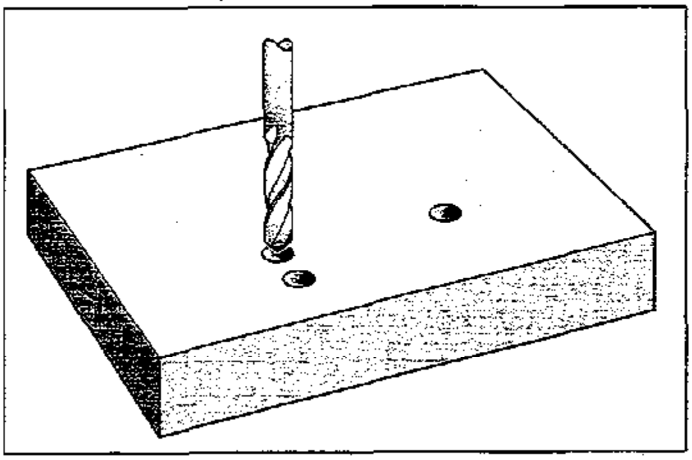


Fig. 8.32: PILOT DRILLING cycle

3. Rough-out

Define and call rough milling tool
G57
Pre-positioning
Cycle call

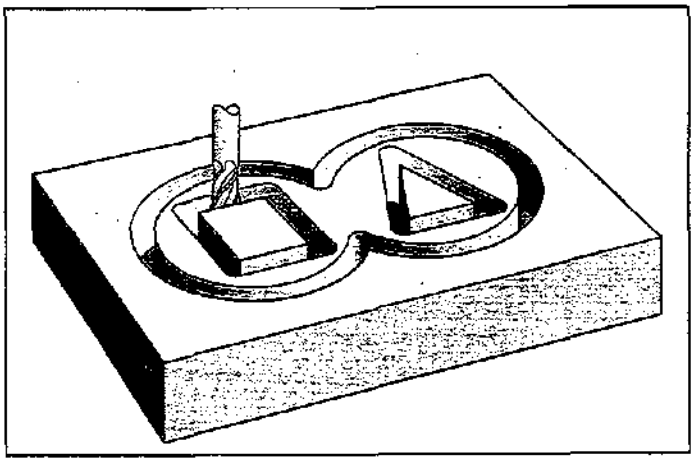


Fig. 8.33: ROUGH-OUT cycle

4. Finishing

Define and call finish milling tool
G58/G59
Pre-positioning
Cycle call

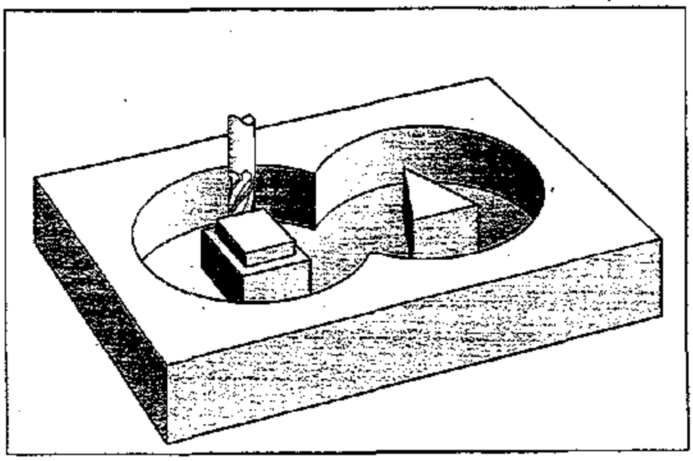


Fig. 8.34: CONTOUR MILLING cycle

5. Contour subprograms

M02 *
Subprograms for the subcontours

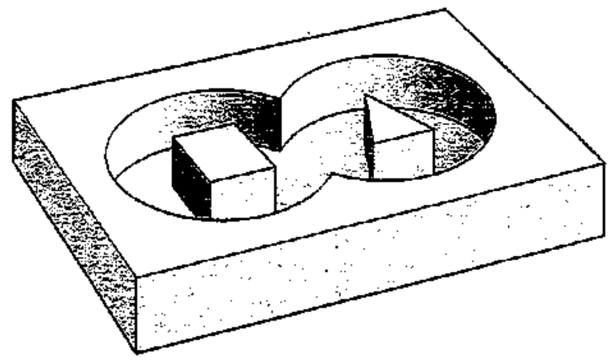
Example: Overlapping pockets with islands

Inside machining with pre-positioning, 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 pages 8-24 and 8-25) 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 pages 8-24 and 8-25
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.

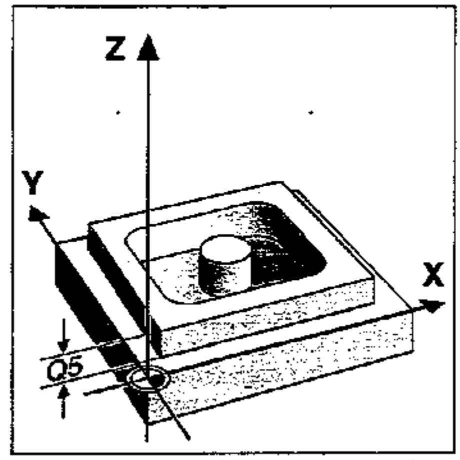


Fig. 8.35: Workpiece surface coordinates Q5

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 COORDINATES Q5
  Absolute coordinates of the workpiece surface referenced to the workpiece 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).
- ROUNDING 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

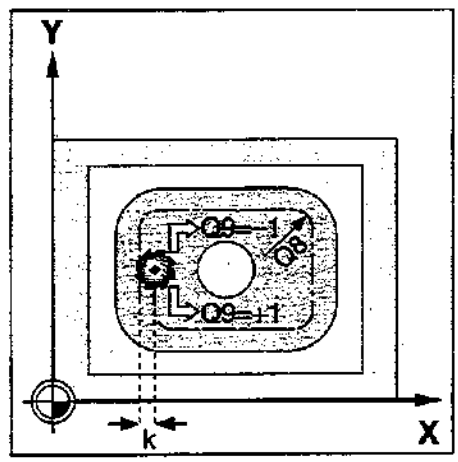


Fig. 8.36: Direction of rotation Q9 and stepover factor k

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 Q14 cannot be used as program parameters.

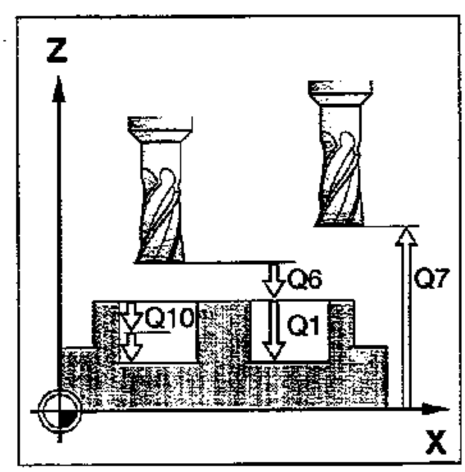


Fig. 8.37: Distance and infeed parameters

PILOT DRILLING (G121)

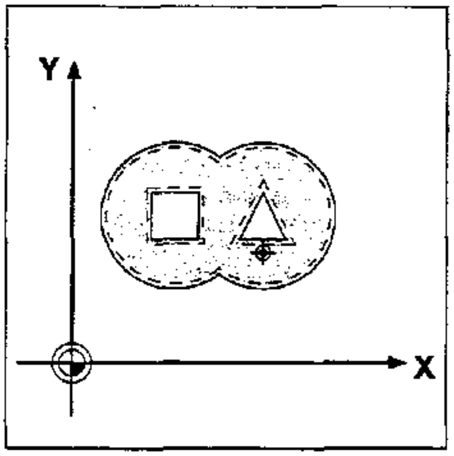


Fig. 8.38: Possible infeed point for PILOT DRILLING

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 MILL Q13
  Tool number of the roughing mill

ROUGH-OUT (G122)

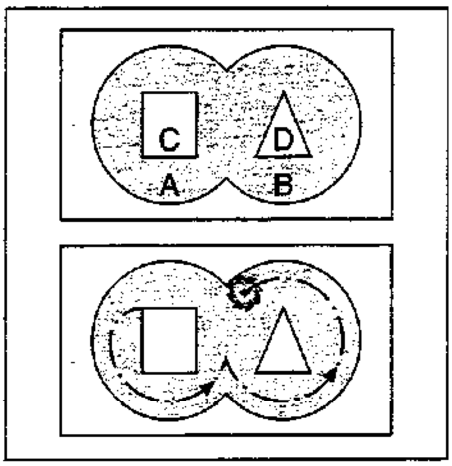


Fig. 8.39: Cutter path for ROUGH-OUT. *A* and *B* are pockets, *C* and *D* are islands

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 8.39) 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) if the pocket is not separately pilot drilled or if the tool must repeatedly jump over contours.

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

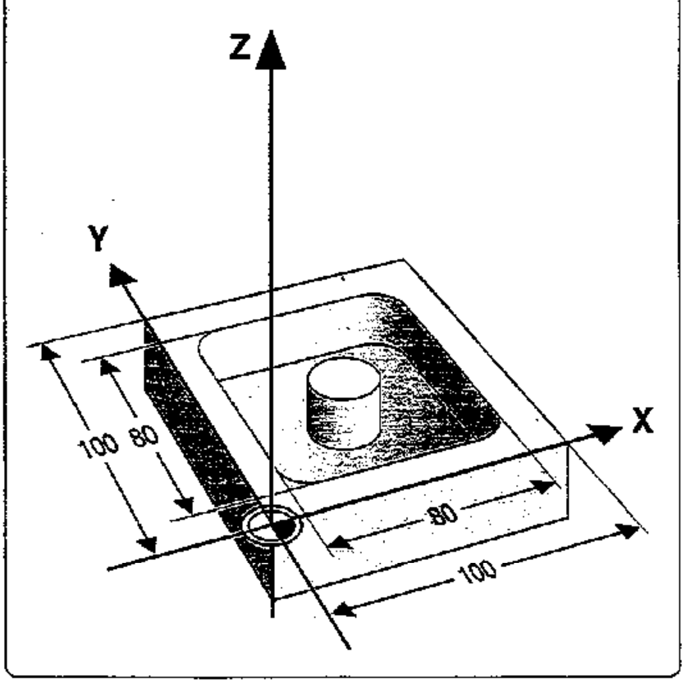
- The sum of ALLOWANCE FOR SIDE (Q14) and the radius of the finish mill must be smaller than sum of ALLOWANCE FOR SIDE (Q3, cycle G120) and the radius of the roughing mill. This calculation also holds if G124 is run without having roughed out with G122, in which case 0 should be used for the radius of 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.



Continued on next page...

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.

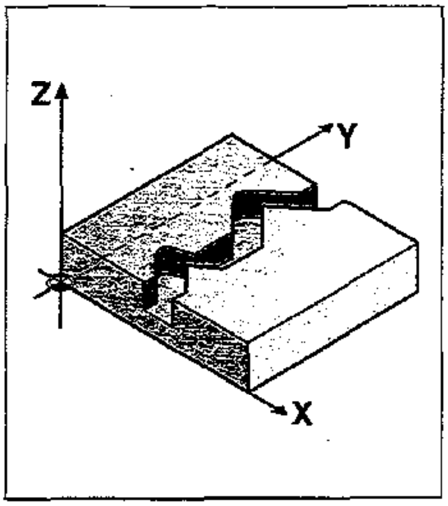


Fig. 8.40: Example of an open contour

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

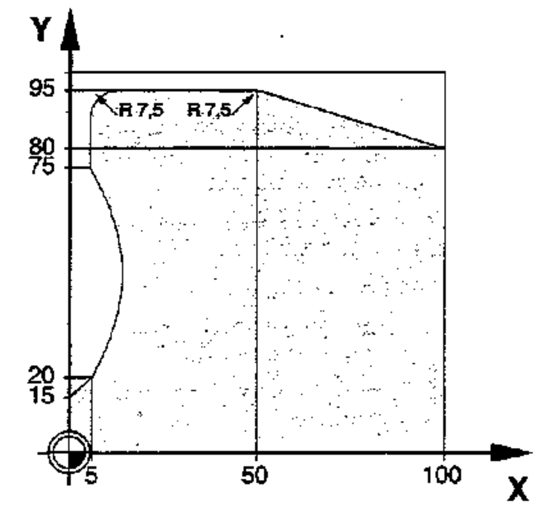
- MILLING DEPTH Q1
  Distance between workpiece surface and contour floor. The sign determines the working direction (a negative sign means negative working direction).
- ALLOWANCE FOR SIDE Q3
  Finishing allowance in the machining plane
- WORKPIECE SURFACE COORDINATES Q5
  Absolute coordinates of the workpiece surface referenced to the workpiece 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 plane
- 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 *
```

8.5 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.

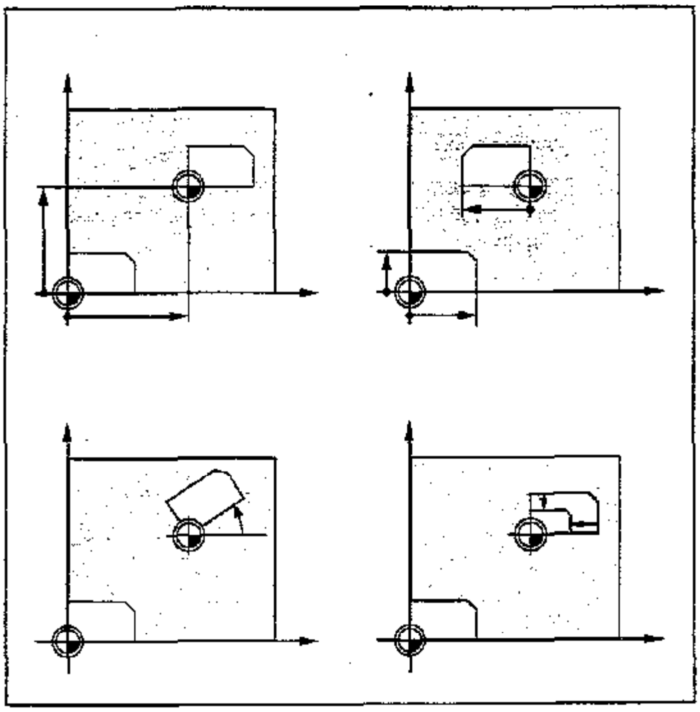


Fig. 8.41: Examples of coordinate transformations

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

DATUM SHIFT (G54)

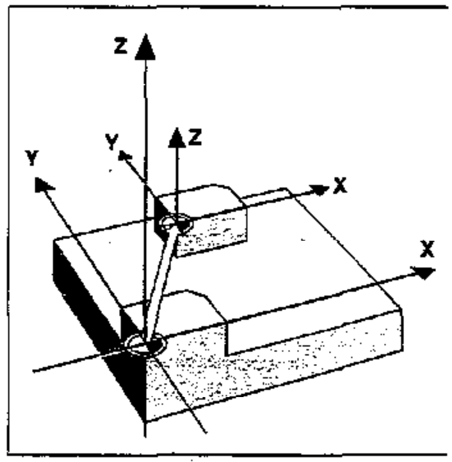


Fig. 8.42: Activation of datum shift

Application

A datum shift allows machining operations to be repeated at various locations on the workpiece.

Activation

After cycle definition of the DATUM SHIFT, 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).

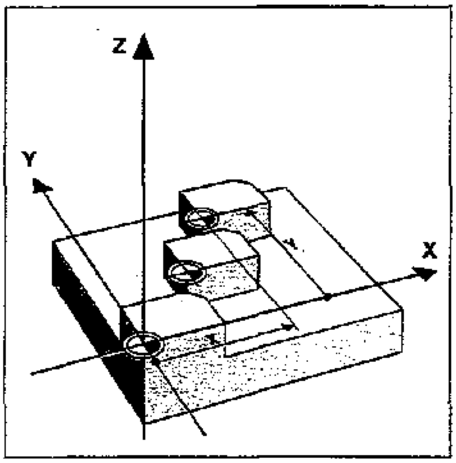


Fig. 8.43: Datum shift, absolute

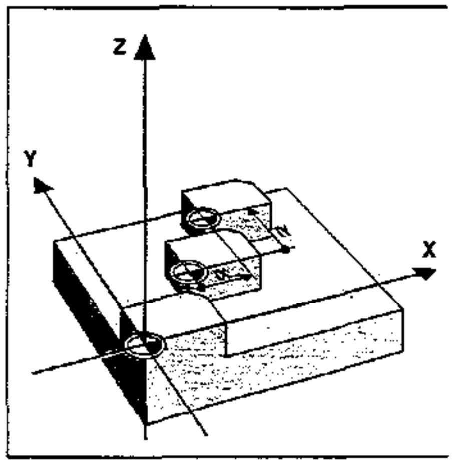


Fig. 8.44: Datum shift, incremental

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.

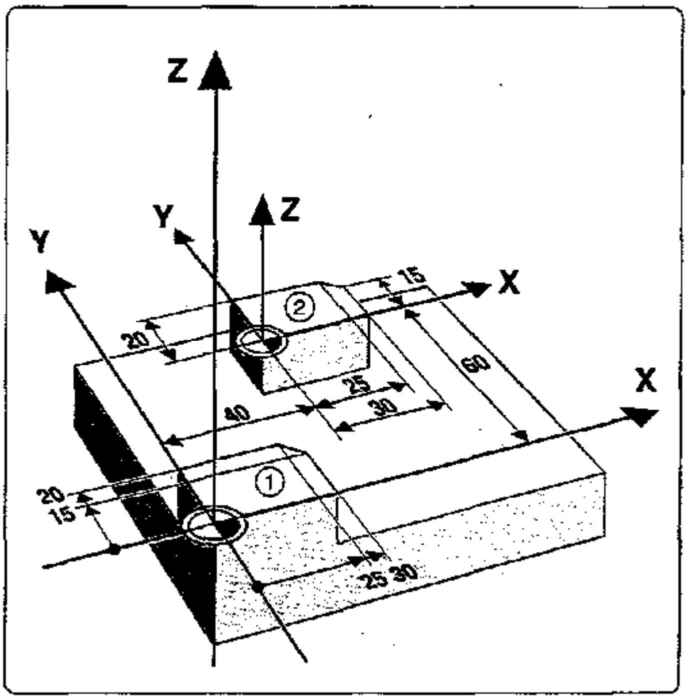
Graphics

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 11-10). 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 subprogram 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.

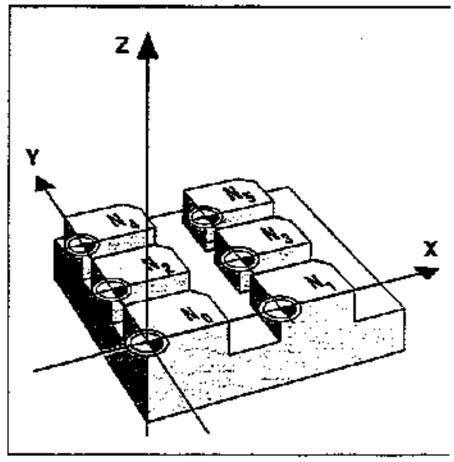


Fig. 8.45: Similar datum shifts

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 8-38).

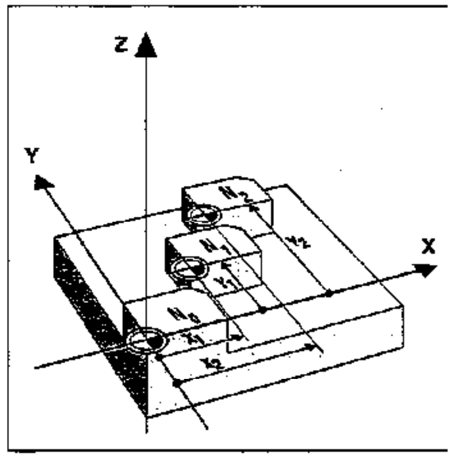


Fig. 8.46: Only absolute datum shifts are possible from a datum table

Editing a datum table

Datum tables are edited in the PROGRAMMING AND EDITING mode:

PGM NAME ▶ 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 beginning of datum table	BEGIN TABLE
• Go to end of datum table	END TABLE
• Page up/down	PAGE ⇧ / PAGE ⇩
• Insert line	INSERT LINE
• Delete line	DELETE LINE
• Enter line, go to beginning of next line	NEXT LINE

- New lines can only be inserted at the end of the file.
- When opening a new datum table, be sure to select the correct dimensions (mm/inch).
- 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 11-15).

MIRROR IMAGE (G28)

Application

This cycle allows you to machine the mirror image of a contour in the machining plane.

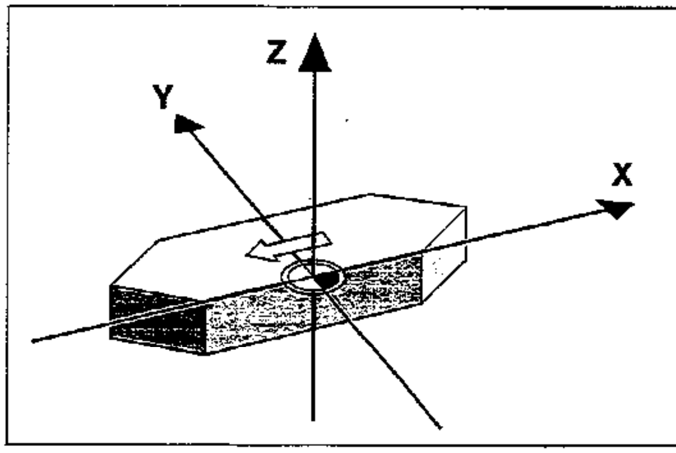


Fig. 8.47: Mirroring a contour

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.

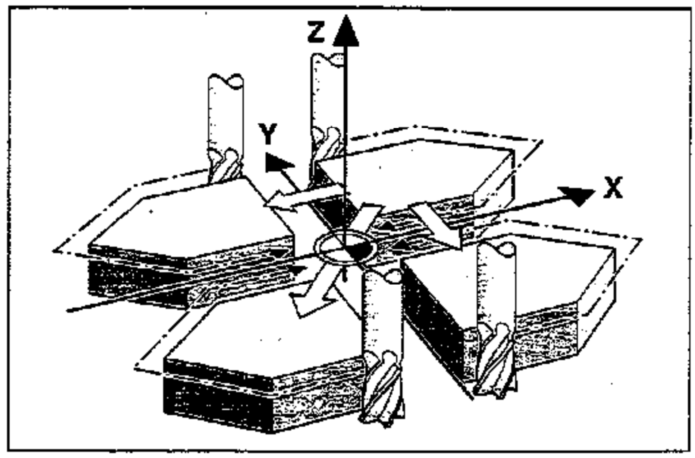


Fig. 8.48: Repeated mirroring, machining direction

Input data

Enter the axes that you wish to mirror. Note that the tool axis cannot be mirrored.

Cancellation

This cycle is cancelled by entering G28 without an axis.

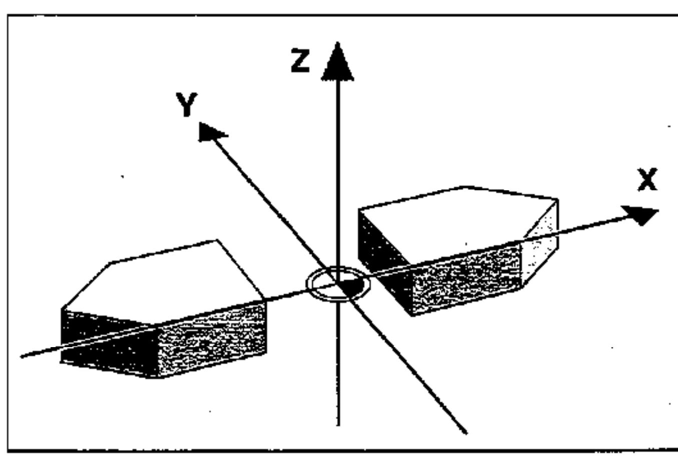
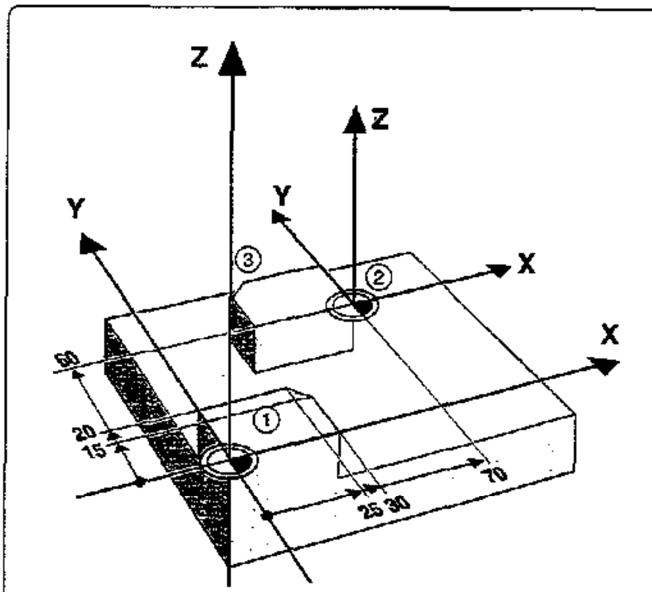


Fig. 8.49: Datum located outside the contour to be mirrored

Example: *Mirror image*

A program section (subprogram 1) is to be executed 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 8-40
  .                                           }
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 immediately upon definition. 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

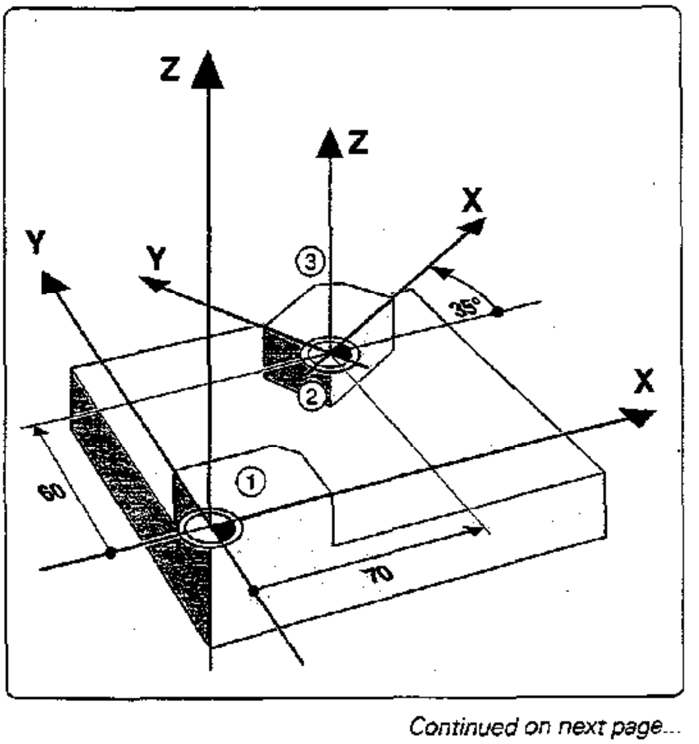
The rotation angle is entered in degrees (°).
Input range: –360° to +360° (absolute or incremental).

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

```
%S846I 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 8-40
      .                        ⎪
N250 G98 L0 *                  ⎭
N99999 %S846I G71 *
```

The corresponding subprogram (see page 8-41) 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

The scaling factor becomes effective immediately upon definition.
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

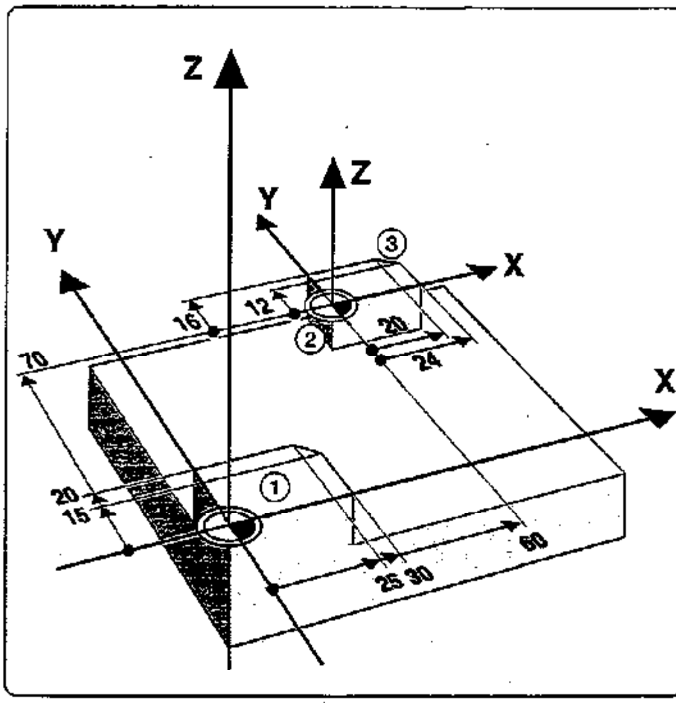
Cancel the scaling factor by entering a scaling factor of 1 in the SCALING FACTOR cycle.

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 8-40
      .                 )
N250 G98 L0 *          )
N99999 %S847I G71 *
```

The corresponding subprogram (see page 8-40) 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)

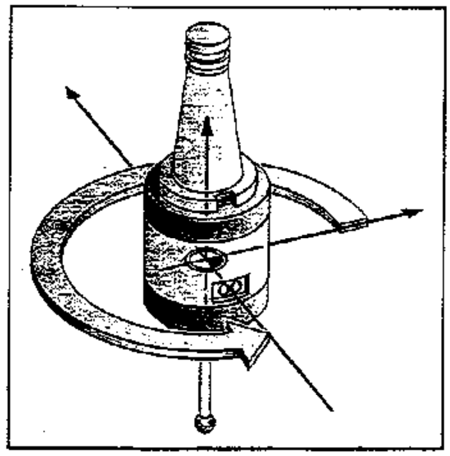


Fig. 8.50: Oriented spindle stop

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.

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°

9 External Data Transfer

The TNC features two interfaces for data transfer between it and other devices.

Application examples:

- Blockwise transfer (DNC mode)
- Reading files into the TNC
- Transferring files from the TNC to external devices
- Printing files

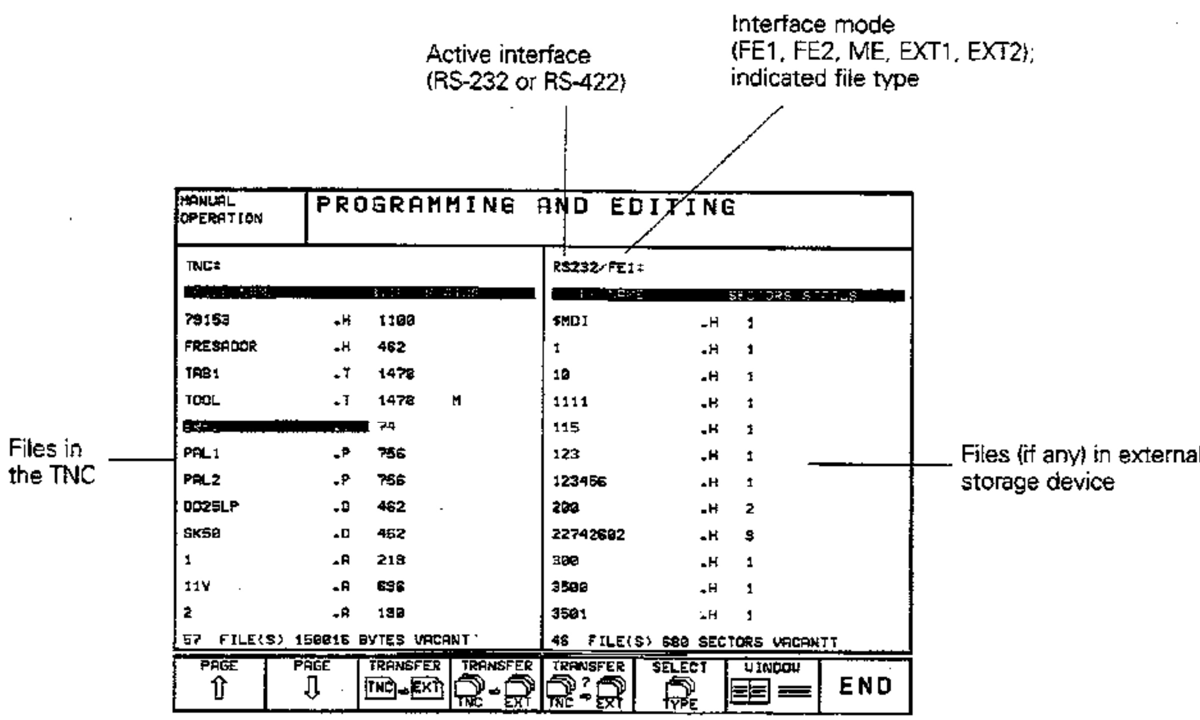
The two interfaces can be used simultaneously.

9.1 Menu for External Data Transfer

To select external data transfer:

	Menu for external data transfer appears on the screen.

The screen is divided into two halves:



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 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:

PAGE ⇧	PAGE ⇩	TRANSFER TNC ⇒ EXT	TRANSFER TNC ⇒ EXT	TRANSFER TNC ? ⇒ EXT	SELECT TYPE	WINDOW	END

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 page 1-27).

Renaming files

Use the soft key RENAME (see page 1-31) to rename files in the TNC, for example when there is already a file in the external device with the same name.

Transferring files

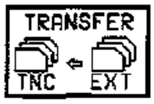
Transferring files from the TNC to an external device

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

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	Key
• Transfer the selected file	
• Transfer all files	
• Select files consecutively for individual transfer. Press ENT to transfer, otherwise press NO ENT	

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.

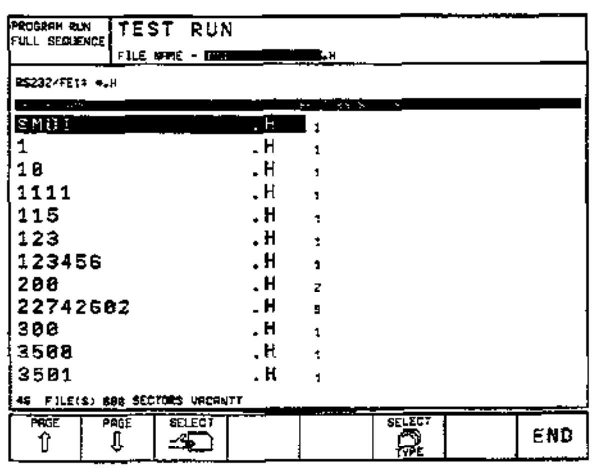


Fig. 9.1: Menu for blockwise transfer

9.3 Pin Layout and Connecting Cable for the Data Interfaces

RS-232-C/V.24 Interface

HEIDENHAIN devices

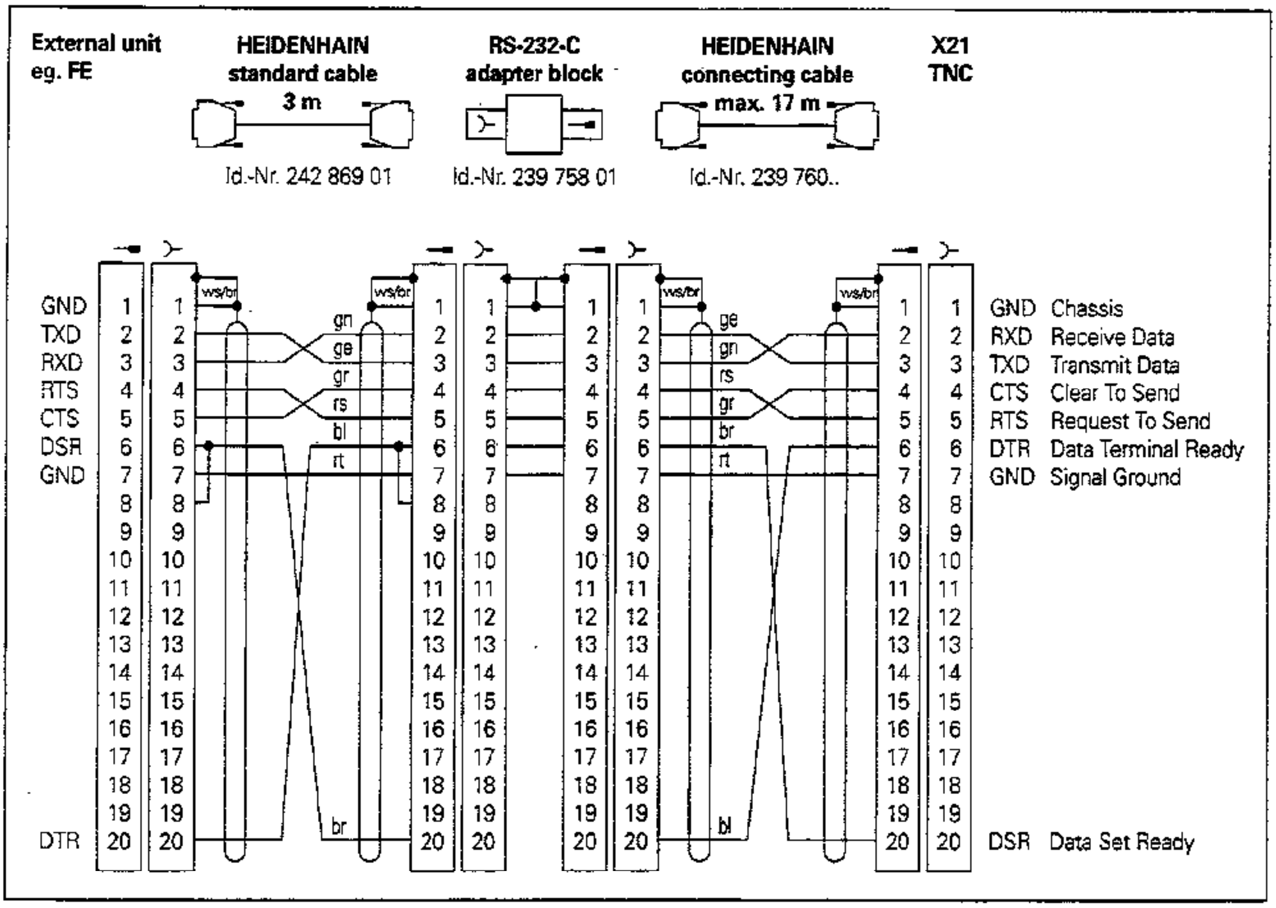


Fig. 9.2: Pin layout of the RS-232-C/V.24 interface for HEIDENHAIN devices

The connector pin layout on the adapter block differs from that on the TNC logic unit (X21).

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.

RS-422/V.11 Interface

Only non-HEIDENHAIN devices are connected to the RS-422 interface.

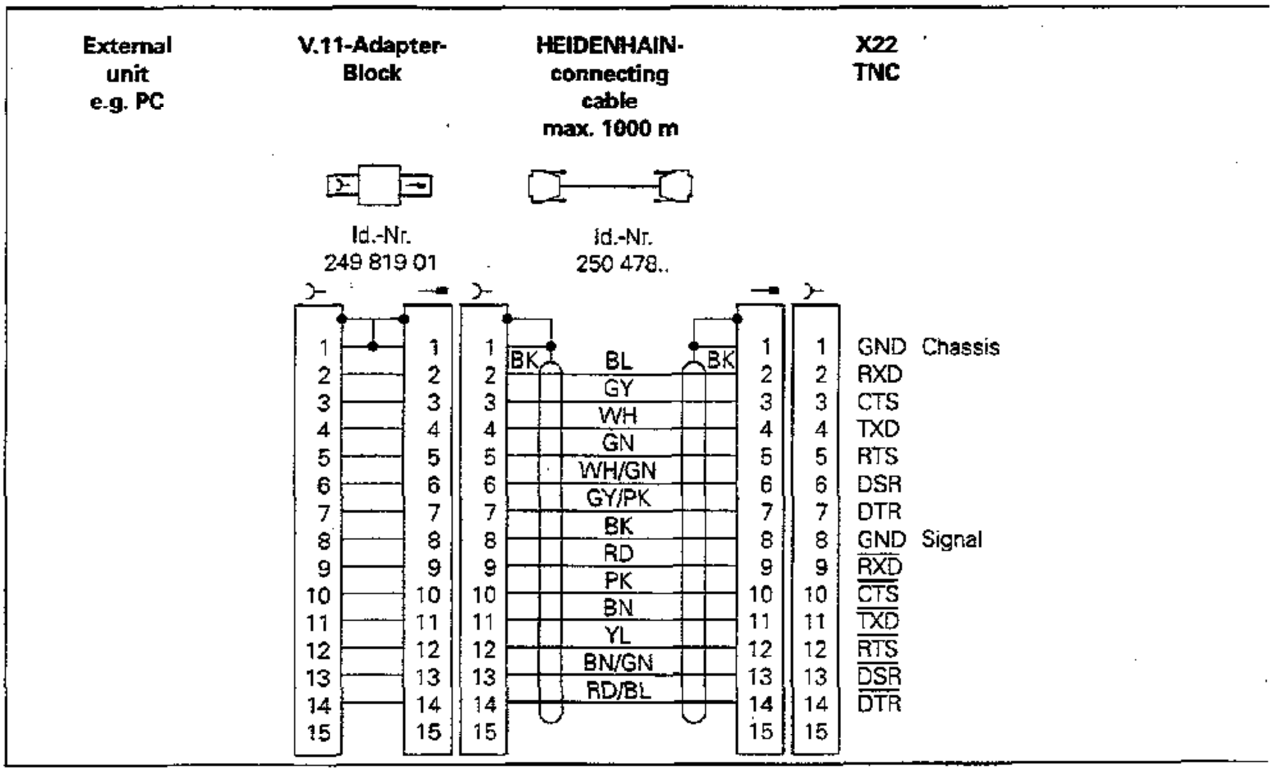


Fig. 9.3: Pin layout of the RS-422/V.11 interface

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 floppy disk into the upper drive
- Format the disk 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 selected at the FE 401.

Non-HEIDENHAIN devices

The TNC and non-HEIDENHAIN devices must be adapted to each other.

Adapting a non-HEIDENHAIN DEVICE to the TNC

- PC: Adapt the software
- Printer: Adjust the DIP switches

Adapting 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).

10 MOD Functions

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)

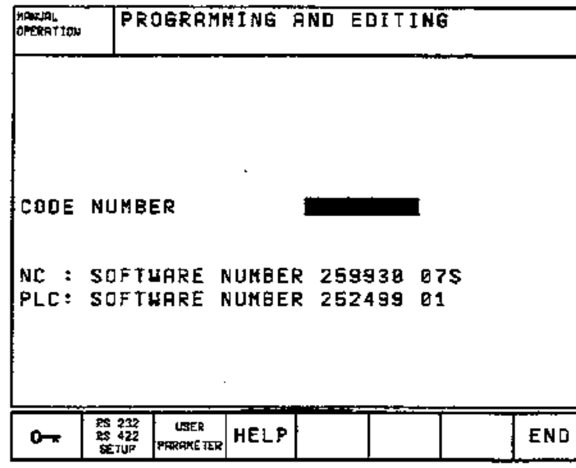


Fig. 10.1: MOD functions in the PROGRAMMING AND EDITING mode

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)

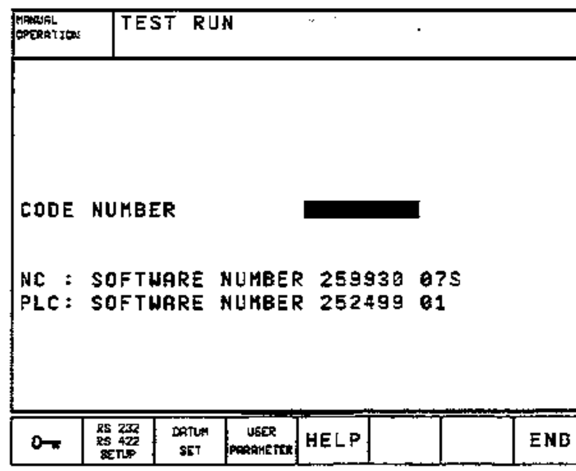


Fig. 10.2: MOD functions in the TEST RUN mode

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)
- Axis traverse limits
- Display datums
- HELP files (if provided)

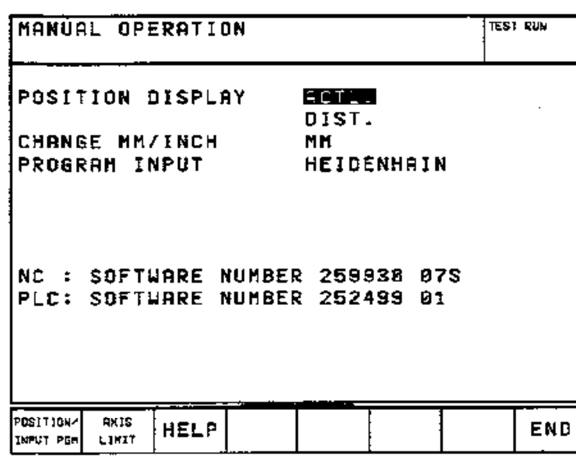


Fig. 10.3: MOD functions in a machine operating mode

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:

END 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:

	Code number
To cancel file erase and edit protection (status P)	86357
To select user parameters	123

10.4 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 • FE 401 with program no. 230 626 03 or higher	FE 1
HEIDENHAIN FE 401 floppy disk unit with program number below 230 626 03 PC with HEIDENHAIN data transfer software TNC.EXE	FE 2
HEIDENHAIN ME 101 magnetic tape unit (no longer produced)	ME
Non-HEIDENHAIN devices such as a 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.

Downward compatibility

For programs that are transferred through the external data interface, the resolution of the numerical data can be set to 0.1 µm or 1 µm.

The 1 µm setting transfers the data with only 3 places after the decimal point in the metric system (4 places in the inch system).

This feature ensures the downward compatibility of the TNC 425 to earlier software versions and other TNCs.

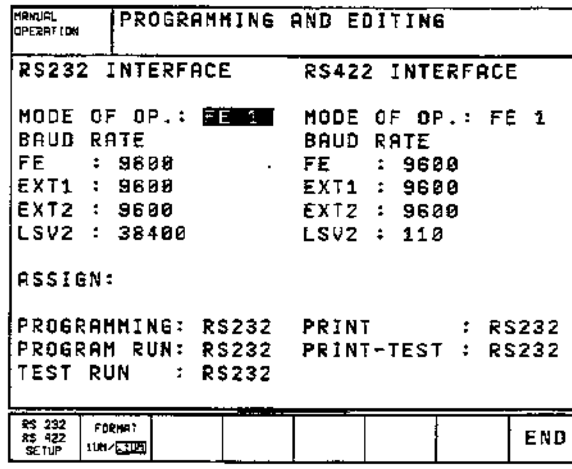


Fig. 10.4: The FORMAT 1 µm / 0.1 µm soft key ensures downward compatibility

Selecting the resolution

To select the resolution of the transferred data, go to the PROGRAMMING AND EDITING mode of operation:

EXT	Call the data transfer menu.
MOD	Select the MOD functions.
FORMAT 1UM/.1UM	Set format to 1 µm or 0.1 µm.
END	Leave the MOD functions.
e.g. TRANSFER TNC ⇒ EXT	Transfer the program.

Setting the baud rate

The baud rate (data transfer speed) can be selected from 110 to 38400 baud.

- The baud rate of the ME 101 is 2400 baud.
- It is not possible to transfer through one interface at 19 200 baud and another interface at 38 400 baud at the same time.

ASSIGN

This function determines which interface (RS-232 or RS-422) is used for external data transfer in the indicated TNC modes of operation.

PRINT and PRINT-TEST

The PRINT and PRINT-TEST functions set the destination for the transferred data.

Applications:
- Transferring values with the Q parameter function FN15
- Transferring digitized surface data

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

PRINT and PRINT-TEST can be set as follows:

Function	Setting
Transfer data via RS-232	RS-232
Transfer data via RS-422	RS-422
Save data to a file in the TNC	FILE
Do not save data	*(Vacant)*

Files in the TNC (FILE setting)

Data	Mode of operation	File name
Digitized data	PROGRAM RUN	Set as in the RANGE cycle
Values with FN15	PROGRAM RUN	% FN15RUN.A
Values with FN15	TEST RUN	%FN15SIM.A

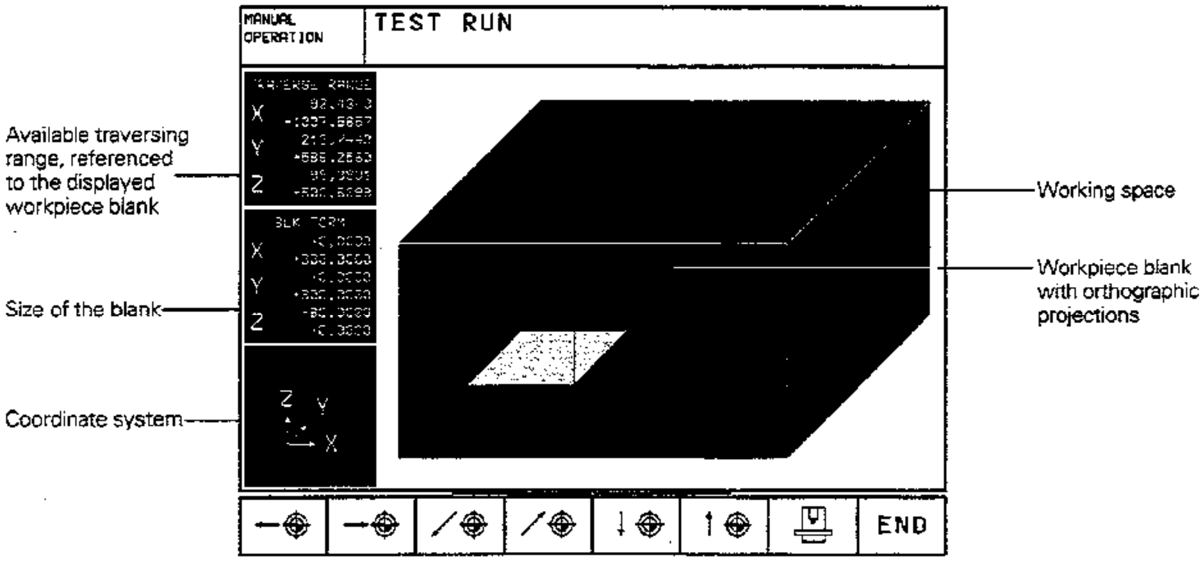
To change a setting, type it into the highlight and confirm by pressing ENT.

10.5 Machine-Specific User Parameters

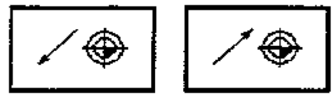
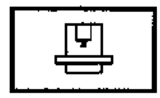
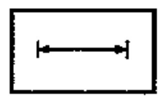
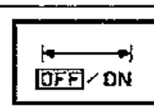
The machine tool builder can assign functions to up to 16 user parameters. For more detailed information on user parameters, refer to your machine operating manual.

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 the work space monitoring in the TEST RUN mode of operation.



Overview of functions

Function	**Soft key**
Move workpiece blank to the left or right (graphically)	
Move the workpiece blank forward or backward (graphically)	
Move the workpiece blank downward or upward (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	
Show a position determined by the machine tool builder (e.g. tool change position) in the working space	
Show the workpiece datum in the working space	
Disable (OFF) or enable (ON) work space monitoring during test run	

10.7 Position Display Types

The positions indicated in figure 10.5 are:

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

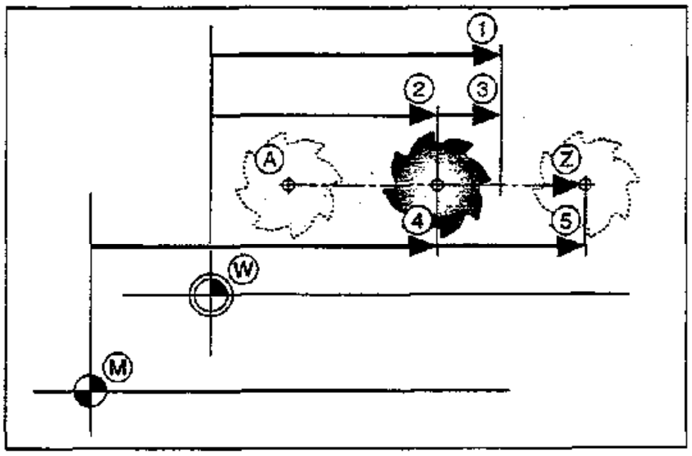


Fig. 10.5: Characteristic positions on the workpiece and scale

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.

The MOD function POSITION DISPLAY (see figure 10.3) permits different types of position infomation for the status display and the additional status display:

- The upper selection
  determines the position display in the status display.
- The lower selection
  determines 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 with 4 digits after the decimal point.

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 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 switch. 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.

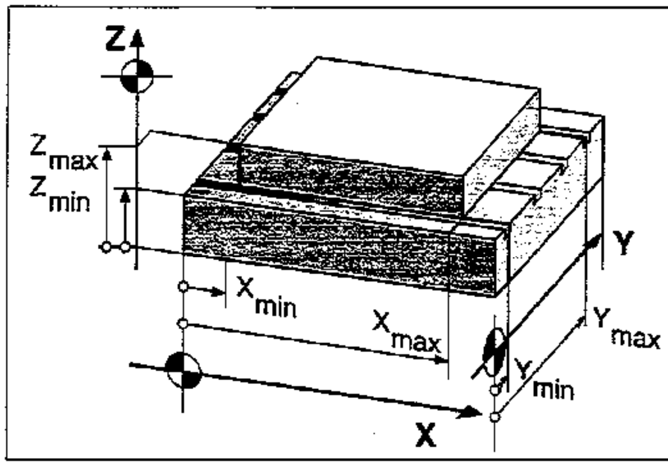


Fig. 10.6: Orienting traverse limits to workpiece size

Working without additional traverse limits

To allow a machine axis to use its full range of traverse in an axis, 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.

	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.

10.11 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.

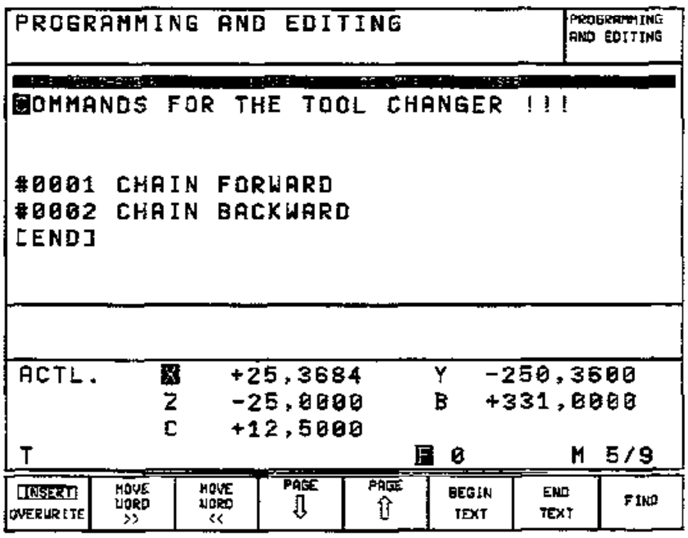


Fig. 10.7 HELP file in a machine operating mode

11 Tables, Overviews and Diagrams

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.

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).

Parameters for external data transfer

Integrating TNC interfaces EXT1 (5020.0) and EXT2 (5020.1) to an external device: data format and transmission stop

Input value: 0 to 255
The input value is the sum of the individual values in the "Value" column.

MP 5020...

Function	Cases	Value
• Number of data bits	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
	BCC control character not permitted	+2
• Transmission stop through RTS	Active	+4
	Inactive	+0
• Transmission stop through DC3	Active	+8
	Inactive	+0
• Character parity	Even	+0
	Odd	+16
• Character parity	Not desired	+0
	Desired	+32
• Number of stop bits	$1\frac{1}{2}$ 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):

MP 5030. ...

Function	Cases	Value
• Interface type	Standard	0
	Interface for blockwise transfer	1

Define the control character for external data transfer

Machine parameters MP 5200 to MP 5210 define ASCII characters as control characters for external data transfer.
Assignment to the interfaces:

EXT 1 MP extension .0
EXT 2 MP extension .1

Input values: ASCII characters 0 to 127

ASCII character for	MP	Value
• Start transmission (STX)	***5200***	ASCII character
• End transmission (ETX)	***5201***	⋮
• Data input (1st character) H	***5202***	⋮
• Data input (2nd character) E	***5203***	⋮
• Data output (1st character) H	***5204***	⋮
• Data output (2nd character) A	***5205***	⋮
• Start of heading (SOH)	***5206***	⋮
• End of transmission block (ETB)	***5207***	⋮
• Positive acknowledgement (ACK)	***5208***	⋮
• Negative acknowledgement (NAK)	***5209***	⋮
• End of transmission (EOT)	***5210***	ASCII character

Parameters for 3D touch probes

Signal transmission for touch probe

MP 6010

Function	Value
• Cable transmission	0
• Infrared transmission	1

Traversing behavior of touch probe

Parameter	Function	Value
MP 6120	Probing feed rate (in mm/min)	80 to 3000
MP 6130	Maximum traverse to the first probe point (mm)	0 to 99 999.999
MP 6140	Safety clearance to probing point during automatic measurement (mm)	0 to 99 999.999
MP 6150	Rapid traverse for probing (mm/min)	1 to 300 000

M function for 180° rotation of the 3D touch probe

The center misalignment of the stylus is compensated with a rotation. The machine tool builder sets the number of the M function that starts the rotation.

MP 6160

Function	Value
• M function active	1 to 88
• M function inactive	0

Reserved machine parameters

The following machine parameters are assigned functions for the HEIDENHAIN measuring touch probe. A description of these functions will be released at some point in the future.

MP	Value
MP 6300	0.1000 to 3.0000
MP 6310	0.100 to 10.000
MP 6320	0 to 7
MP 6330	0.1000 to 4.0000
MP 6340	0.0001 to 0.5000
MP 6350	80 to 3000
MP 6360	80 to 3000
MP 6370	0.0000 to 10.0000
MP 6380	0.000 to 10.000

Parameters for TNC displays and the editor

Programming station

MP 7210

Function	Value
• TNC with machine	0
• TNC as programming station with active PLC	1
• TNC as programming station with inactive PLC	2

Automatic acknowledgment of POWER INTERRUPTED message

MP 7212

Function	Value
• Acknowledge power interruption with key	0
• Power interruption automatically acknowledged	1

Block number increment for ISO programming

MP 7220

Function	Value
• Block number increment	0 to 150

Length of file names

MP 7222

Function	Value
• File names with maximum 8 characters	0
• File names with maximum 12 characters	1
• File names with maximum 16 characters	2

Inhibiting file management for particular file types

Input value: 0 to 63 (sum of the individual values in the "Value" column).

If you do not wish to inhibit file management for a particular file type, use the value 0.

If the file management function is inhibited for existing files, these files will be erased.

MP 7224.0

Inhibit file management for	Value
• HEIDENHAIN programs	+1
• ISO programs	+2
• Tool tables	+4
• Datum tables	+8
• Pallet tables	+16
• Text files	+32

Inhibiting the editor for certain file types

Input value: 0 to 63 (sum of the individual values in the "Value" column).

If you do not wish to inhibit the editor for a particular file type, use the value 0.

MP 7224.1

Inhibit editor for	Value
• HEIDENHAIN programs	+1
• ISO programs	+2
• Tool tables	+4
• Datum tables	+8
• Pallet tables	+16
• Text files	+32

Activating tables

If you do not want to activate any tables, enter 0

Parameter	Function	Value
• ***MP 7226.0***	Number of pallets per pallet file	0 to 255
• ***MP 7226.1***	Number of datums per datum table	0 to 255
• ***MP 7260***	Number of tools per tool table	0 to 254
• ***MP 7261***	Number of pockets per pocket table	0 to 254

Making a tool and pocket table

Tool name – NAME:	***MP 7266.0***	Tool number – T:	***MP 7267.0***
Tool length – L:	***MP 7266.1***	Special tool – ST:	***MP 7267.1***
Tool radius – R:	***MP 7266.2***	Fixed pocket – F:	***MP 7267.2***
Tool radius – R2	***MP 7266.3***	Pocket locked – L:	***MP 7267.3***
Oversize length – DL:	***MP 7266.4***	PLC – Status – PLC:	***MP 7267.4***
Oversize radius – DR:	***MP 7266.5***		
Oversize radius 2 – DR2:	***MP 7266.6***		
Tool locked – TL:	***MP 7266.7***		
Replacement tool – RT:	***MP 7266.8***		
Maximum tool life – TIME1:	***MP 7266.9***		
Maximum tool life for TOOL CALL – TIME2:	***MP 7266.10***		
Current tool age – CUR. TIME:	***MP 7266.11***		
Tool comment – DOC:	***MP 7266.12***		

Function	Value
• Column number of the data in the tool table	1 to 13
• Column number of the data in the pocket table	1 to 5
• Do not show data in the table	0

Dialog language

MP 7230

Function	**Value**
• National language	0
• English (standard)	1

Protect OEM cycles

This parameter prevents the editing of any program whose name is the number of a machine manufacturer cycle (OEM cycle).

MP 7240

Function	**Value**
• Protect OEM cycles	0
• Do not protect OEM cycles	1

Feed rate display in the MANUAL OPERATION mode of operation

MP 7270

Function	**Value**
• Display "F=0" if one axis direction button is pressed; Display "F" (without value) if more than one axis direction button is pressed	0
• Display the feed rate of the slowest axis, regardless of the number of axis direction keys pressed	1

Decimal character

MP 7280

Function	**Value**
• The decimal character is a point	1
• The decimal character is a comma	0

Tool length in the coordinate display

MP 7285

Function	**Value**
• Display the position of the tool datum	0
• Display the position of the tool face	1

Display steps for coordinate axes

X axis: ***MP 7290.0***
Y axis: ***MP 7290.1***
Z axis: ***MP 7290.2***
IV axis: ***MP 7290.3***
V axis: ***MP 7290.4***

MP 7290

Function	Value
• Display step 0.1 mm	0
• Display step 0.05 mm	1
• Display step 0.01 mm	2
• Display step 0.005 mm	3
• Display step 0.001 mm	4
• Display step 0.0005 mm	5
• Display step 0.0001 mm (TNC 425 only)	6

Inhibit datum setting

Input value: 0 to 31 (sum of values in the "Value" column).

If you do not want to inhibit a given axis for datum setting, the value for that axis is 0:

If datum setting is inhibited for all axes, the TNC removes the DATUM SET soft key in the MANUAL OPERATION mode.

MP 7295

Function	Value
Inhibit datum setting for X axis	+1
Inhibit datum setting for Y axis	+2
Inhibit datum setting for Z axis	+4
Inhibit datum setting for axis IV	+8
Inhibit datum setting for axis V	+16

MP 7296

Function		Value
Set datum only with soft key	DATUM SET	1
Set datum with soft key or with orange axis key	DATUM SET	0

Erase the status display, Q parameters and tool data after program run

The status display and the Q parameters can be erased at the end of the program with a PGM END block, M2 or M30.

MP 7300

Function	Value
• Erase status display, Q parameters and tool data when a program is selected	0
• Erase status display, Q parameters and tool data with M02, M30, END PGM and when a program is selected	1
• Erase status display and tool data when a program is selected	2
• Erase status display and tool data when a program is selected and with M02, M30, and END PGM	3
• Erase status display and Q parameters when a program is selected	4
• Erase status display, Q parameters and tool data with M02, M30, END PGM and when a program is selected	5
• Erase status display when a program is selected	6
• Erase status display with M02, M30, END PGM and when a program is selected	7

Graphic display mode

Input value: 0 to 15 (sum of values in the "Value" column)

MP 7310

Function	Cases	Value
• Projection in three planes according to ISO 6433	Projection method 1	+0
	Projection method 2	+1
• Rotate coordinate system by 90°	Rotate	+2
	Do not rotate	+0
• Shift the new BLK FORM with cycle 7 DATUM SHIFT (see page 8 ...)	Shift	+4
	Do not shift	+0
• Show cursor position during "projection in 3 planes" mode	Show	+8
	Do not show	+0

Graphic simulation without programmed tool axis

Enter any realistic value

Parameter	Function	Value
• ***MP 7315***	Tool radius	+0
• ***MP 7316***	Penetration depth from upper surface of blank form	+2
• ***MP 7317.0***	M function for starting graphic simulation	+4
• ***MP 7317.1***	M function for ending graphic simulation	+8

Parameters for machining and program run

Oriented spindle stop with cycle G85

MP 7160

Function	Value
• Spindle orientation at beginning of cycle G85	0
• No spindle orientation at beginning of cycle G85	1

Size of NC memory for blockwise transfer

MP 7228

Function		Value
• MP 7228.0	Minimum memory range (sectors)	1–1024
• MP 7228.1	Maximum memory range (sectors)	1–1024

One sector is approximately 1 kilobyte.

Effect of cycle G72 SCALING FACTOR

MP 7410

Function	Value
• SCALING effective in 3 axes	0
• SCALING effective in the working plane	1

Tool compensation data in the TOUCH PROBE block

MP 7411

Function	Value
• Current tool data are overwritten by the calibrated data from the touch probe system	0
• Current tool data are retained	1

Behavior of machining cycles

This general user parameter affects the pocket milling technique.

Input value: 0 to 15 (sum of the individual values in the "Value" column).

MP 7420

Function	Cases	Value
• Direction for milling a channel around the contour	Clockwise for pockets, counterclockwise for islands	+1
	Counterclockwise for pockets, clockwise for islands	+0
• Sequence of roughing-out and channel milling	First mill the channel, then rough-out the pocket	+0
	First rough-out the pocket, then mill the channel	+2
• Combining contours	Combine compensated contours	+0
	Combine uncompensated contours	+4
• Milling in depth	Mill the channel and rough-out for each infeed depth before continuing to the next depth	+8
	Complete one process for all infeeds before switching to the other process	+0

Overlap factor for pocket milling

Amount of overlap for pocket milling:
Product of MP 7430 and the tool radius

MP 7430

Function	Value
• Overlap factor for pockets	0.1 to 1.414

Circular path tolerance

This parameter sets the distance by which a programmed end point can be removed from the path of a perfect circle.

MP 7431

Function	Value
• Circular path tolerance (mm)	0.0001 to 0.016

Behavior of M functions

Input value: 0 to 31 (sum of the values in the "Value" column)

MP 7440

Function	Cases	Value
• Programmable stop with M6	Program stop	+0
	No program stop	+1
• Modal cycle call at end of block through M89	Cycle call	+2
	No cycle call	+0
• Program stop with M functions	Program stop	+0
	No program stop	+4
• Switching the Kv factor through M105 and M106	Kv factor can be switched	+8
	Kv factor cannot be switched	+0
• Reduce the feed rate in the tool axis with M103 F...	Function not effective	+0
	Function effective	+16

The Kv factors for position loop gain are set by the machine tool builder. He can give you more detailed information on this subject.

Safety limit for machining corners at a constant feed rate

A corner whose angle is less than the entered value will be machined at a reduced feed rate if radius compensation is R0 or if the angle is at an inside corner.

This feature does not work during operation with servo lag or feed precontrol.

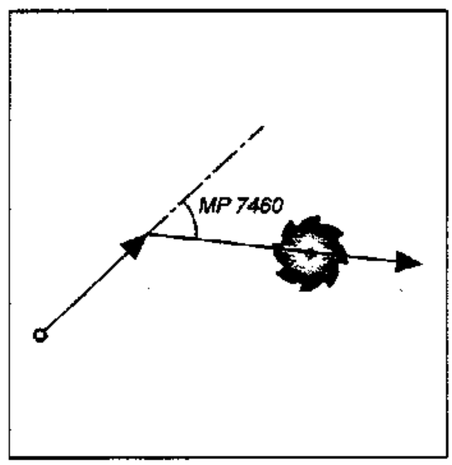


Fig. 11.1: Sharpest permissible angle for constant contouring speed

MP 7460

Function	Value
• Constant feed rate in corners for inside angles (in degrees)	0.0000 to 179.9999

Coordinate system for datums from a datum table

MP 7475

Function	Value
• Datums from a table are referenced to the workpiece datum	0
• Datums from a table are referenced to the machine datum	1

Parameters for the electronic handwheel

Setting the TNC for handwheel operation

Input value: 0 to 5

MP 7640

Function	Value
• No 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 keys for traverse direction and rapid traverse are evaluated by the PLC	3
• HR 332 with twelve additional keys	4
• Fixed-axis handwheels with additional keys	5

Interpolation factor

MP 7641

Function	Value
Interpolation factor is entered at the keyboard	0
Interpolation factor is set by the PLC	1

Initializing the handwheel

This machine parameter reserves 8 bytes for initializing a handwheel.

Input value: 0 to 255

MP 7645.x (MP 7645.0* to *MP 7645.7)

Function
The machine-tool builder sets the functions of the individual machine parameters for the handwheel.

11.2 Miscellaneous Functions (M Functions)

Miscellaneous functions with predetermined effect

	Function	Effective at	
		Start of block	**End of block**
M00	Stop program run/spindle STOP/coolant OFF		•
M02	Stop program run/spindle STOP/coolant OFF/clear status display (depending on machine parameter)/go to block 1		•
M03	Spindle ON clockwise	•	
M04	Spindle ON counterclockwise	•	
M05	Spindle STOP		•
M06	Tool change/stop program run (depending on machine parameter)/ Spindle STOP		•
M08	Coolant ON	•	
M09	Coolant OFF		•
M13	Spindle ON clockwise/coolant ON	•	
M14	Spindle ON counterclockwise/coolant ON	•	
M30	Same as M02		•
M89	Vacant miscellaneous function	•	
	— or —		
	Cycle call, modally effective (depending on machine parameter)		•
M90	Constant contouring speed at corners (effective only in lag mode)	•	
M91	**Within the positioning block:** Coordinates are referenced to machine datum	•	
M92	**Within the positioning block:** Coordinates are referenced to position defined by machine builder, such as tool change position	•	
M93	Reserved	•	
M94	Reduce display of rotary axis to value less than 360°	•	
M95	Reserved		•
M96	Reserved		•
M97	Machine small contour steps		•
M98	Completely machine open contours		•
M99	Blockwise cycle call		•
M101	Automatic tool change with sister tool if maximum tool life has expired	•	•
M102	Reset M101		
M103	Reduce feed rate during plunging to factor F (percentage)	•	
M105	Machining with first Kv factor	•	
M106	Machining with second Kv factor	•	
M107	Suppress error message for sister tools with oversize	•	
M108	Reset M107		•
M109	Constant contouring speed at tool cutting edge on circular arcs (increase and decrease feed rate)	•	
M110	Constant contouring speed at circular arcs (feed rate decrease only)	•	
M111	Reset M109/M10		•
M112	Automatic insertion of rounding arcs at non-tangential straight-line transitions; Enter tolerance T for contour deviation	•	
M113	Reset M112		•
M114	Automatic compensation of machine geometry during operation with tilting axes	•	
M115	Reset M114		•
M116	Feed rate for angular axes in mm/min	•	
M118	Superimpose handwheel positioning during program run	•	

The miscellaneous functions M105 and M106 are defined and enabled by the machine builder. Please contact your machine builder for more information.

Vacant miscellaneous functions

The vacant miscellaneous functions are used by the machine tool builder for machine-specific functions. You will find a description of these functions in the operating manual for your machine tool.

Effect of vacant miscellaneous functions

	Function	Effective at start of block	Effective at end of block
M01			•
M07		•	
M10			•
M11		•	
M12			•
M15		•	
M16		•	
M17		•	
M18		•	
M19			•
M20		•	
M21		•	
M22		•	
M23		•	
M24		•	
M25		•	
M26		•	
M27		•	
M28		•	
M29		•	
M31		•	
M32			•
M32			•
M34			•
M35			•
M36		•	
M37		•	
M38		•	
M39		•	
M40		•	
M41		•	
M42		•	
M43		•	
M44		•	
M45		•	
M46		•	
M47		•	
M48		•	
M49		•	

	Function	Effective at start of block	Effective at end of block
M50		•	
M51		•	
M52			•
M53			•
M54			•
M55		•	
M56		•	
M57		•	
M58		•	
M59		•	
M60			•
M61		•	
M62		•	
M63			•
M64			•
M65			•
M66			•
M67			•
M68			•
M69			•
M70			•
M71		•	
M72		•	
M73		•	
M74		•	
M75		•	
M76		•	
M77		•	
M78		•	
M79		•	
M80		•	
M81		•	
M82		•	
M83		•	
M84		•	
M85		•	
M86		•	
M87		•	
M88		•	

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 PGM CALL) is programmed in millimeters or inches.

Dimensions of the main program	Parameter value
Metric system (mm) Inch system	Q113 = 0 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 a 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

11.4 Diagrams for Machining

Spindle speed S

The spindle speed *S* can be calculated from the tool radius *R* and the cutting speed *V* as follows:

$$S = \frac{V}{2\pi R}$$

Units:

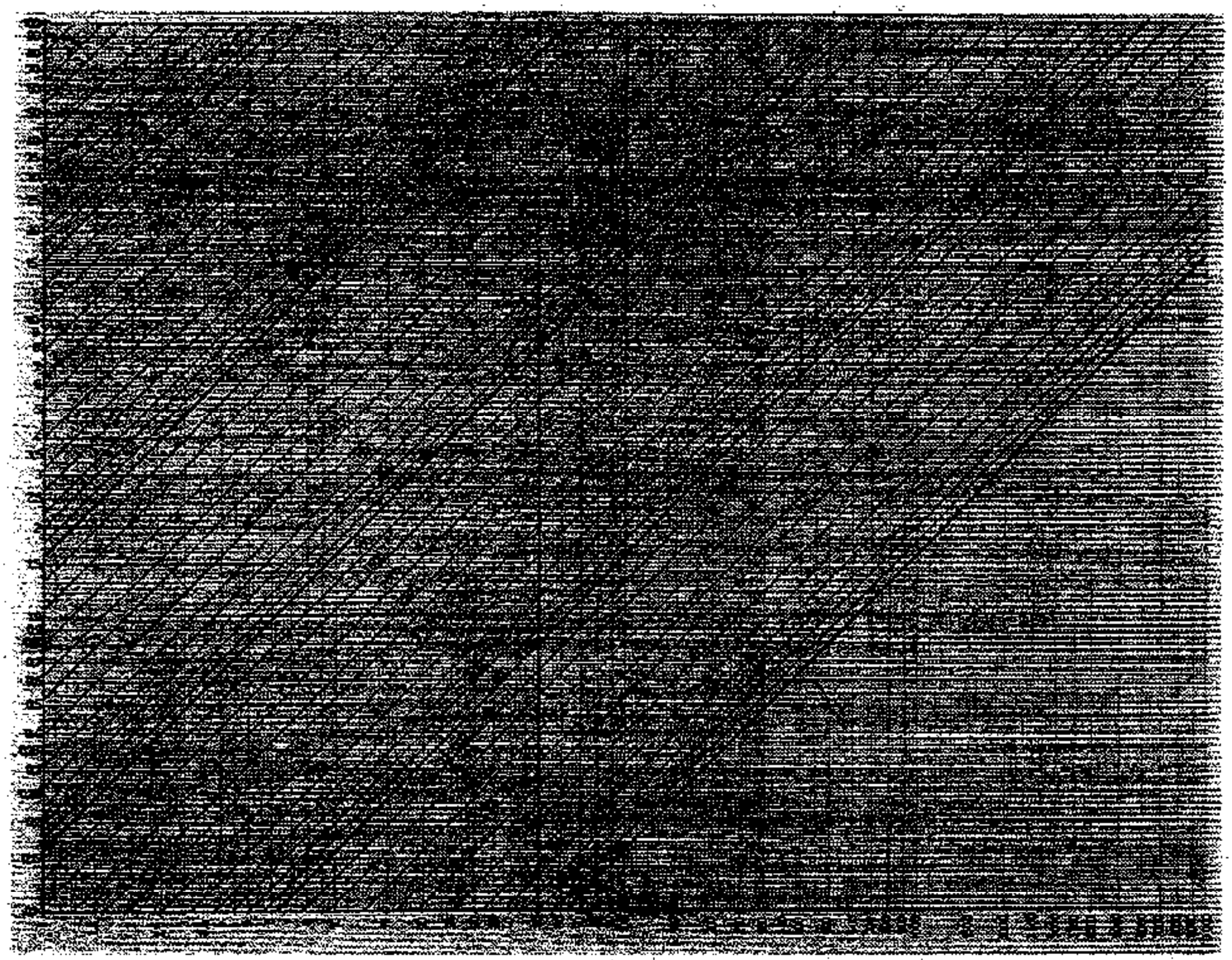
S in rpm
V in m/min
R in mm

You can either read the spindle speed directly off the diagram below or calculate it with the above formula.

Example:

Tool radius	R = 15 mm
Cutting velocity	V = 50 m/min
Spindle speed	S = 500 rpm
	(calculated S = 530 rpm)

Tool radius
R [mm]



Cutting velocity
V [m/min]

Feed rate F

The feed rate of the tool *F* is calculated from the number of tool teeth *n*, the permissible depth of cut per tooth *d* and the spindle speed *S*:

$F = n \cdot d \cdot S$

Units:

F in mm/min
d in mm
S in rpm

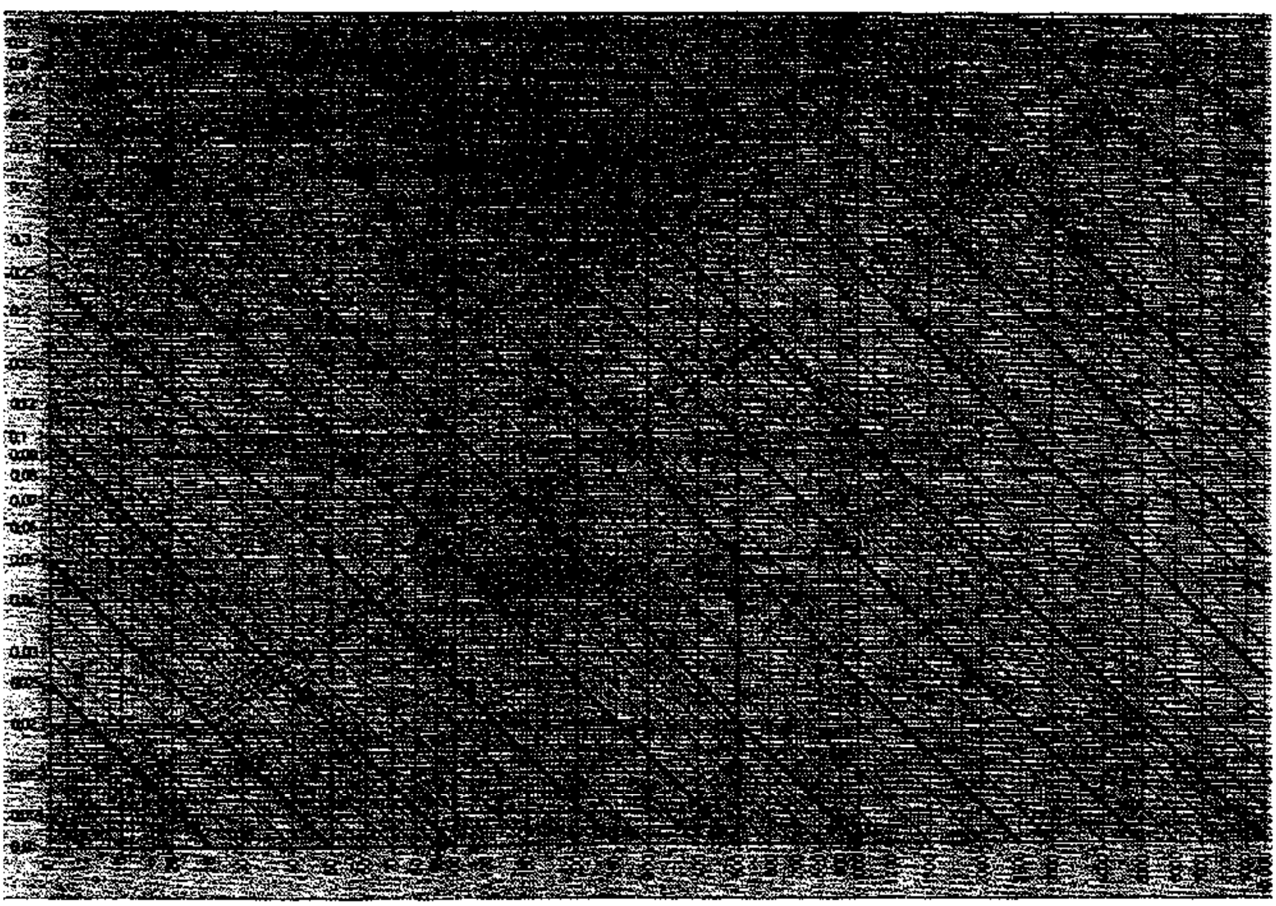
The feed rate that is read from the diagram must be multiplied by the number of tool teeth.

Example:

Depth of cut per tooth	d	=	0.1 mm
Spindle speed	S	=	500 rpm
Feed rate from diagram	F	=	50 mm/min
Number of tool teeth	n	=	6
Feed rate to enter	F	=	300 mm/min

This diagram provides a useful approximation of the values resulting from this calculation. It assumes the following:
- Downfeed of the tool axis is = 0.5 · R and the tool is cutting through solid metal, or
- Lateral metal-to-air ratio = 0.25 · R and the downfeed in the tool axis = R

Depth of cut per tooth
d [mm]



Spindle speed
S [rpm]

Feed rate for tapping F

The feed rate for tapping *F* is calculated from the thread pitch *p* and the spindle speed *S*:

$F = p \cdot S$

Units:

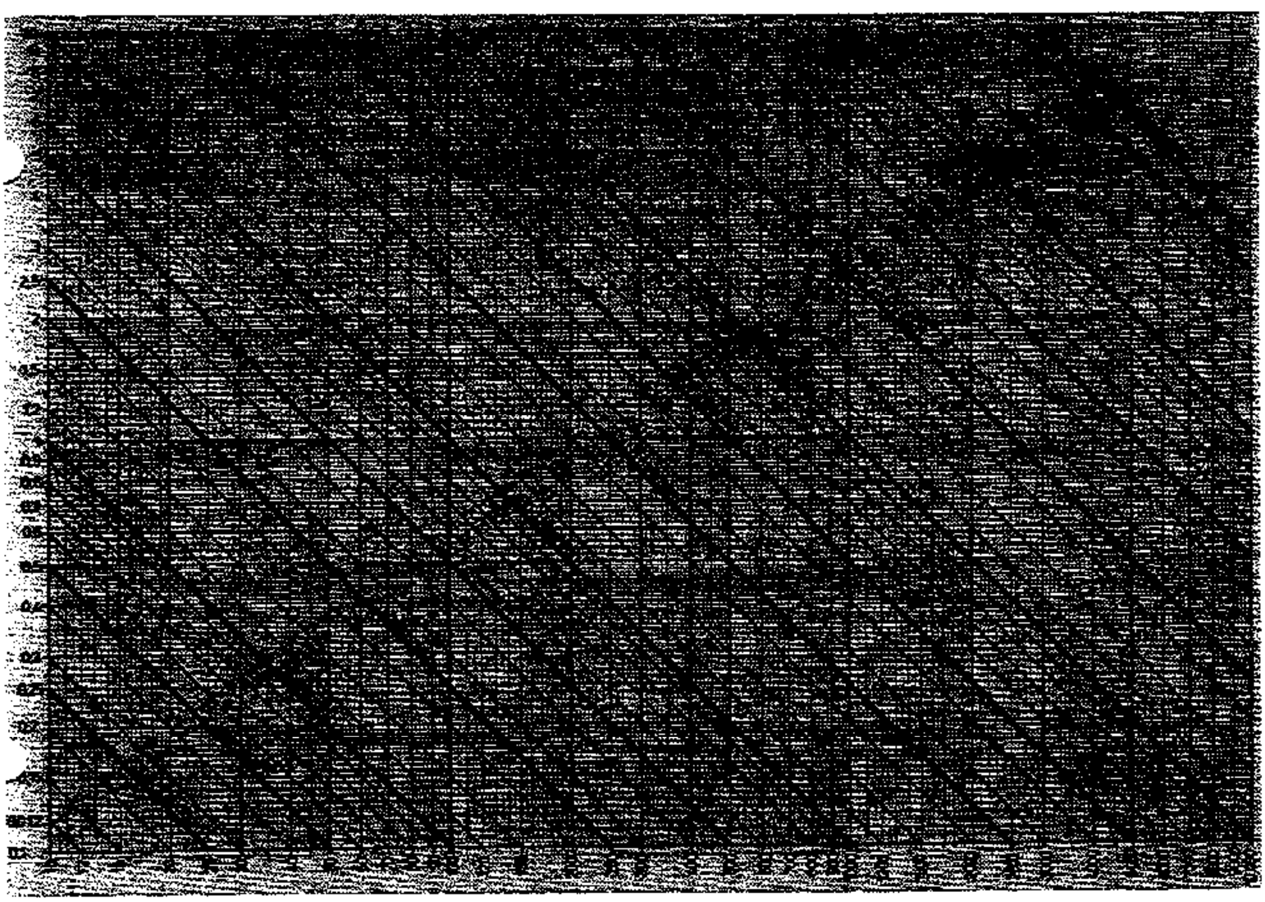
F in mm/min
p in mm/1
S in 1/mm

The feed rate for tapping can be read directly from the diagram below.

Example:

Thread pitch	p	=	1 mm/rev
Spindle speed	S	=	100 rpm
Feed rate for tapping	F	=	100 mm/min

Thread pitch
p [mm/rev]



Spindle speed
S [rpm]

11.5 Features, Specifications and Accessories

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 HEIDENHAIN software TNC REMOTE.

Simultaneous axis control for contour elements

- Straight lines: up to 5 axes
  (TNC 407: 3 axes;
  export versions TNC 415 F and TNC 425 E: 4 axes)
- Circles: up to 3 axes (with tilted working plane)
- Helices: 3 axes

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 and ISO programming
- Tool tables, datum tables, pallet files
- Text and system files

Program memory

- Battery-buffered for up to 100 files
- Capacity 256K bytes (TNC 407: 128K bytes)

Tool definitions

- Up to 254 tools in the program or in tables

"Look Ahead"

- Defined rounding of discontinuous contour transitions (such as for 3D surfaces)
- Collision prevention with the SL cycle for open contours
- Geometry pre-calculation for feed rate adaptation

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 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, tapping (also with synchronized spindle), thread cutting, rectangular and circular pocket milling, slot milling, milling pockets 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 digitizing 3D surfaces (optional)

Mathematical functions

Basic operations +, −, x, ÷
Trigonometric functions sine, cosine, tangent, arc sine, arc cosine, arc tangent
Square root of values ($\sqrt{a}$) and root sum of squares ($\sqrt{a^2 + b^2}$)
Squaring (SQ)
Square roots (^)
Negation (NEG)
Forming an absolute number (ABS)
Forming an integer (INT)
Dropping the values before the decimal point (FRAC)
Comparisons (greater than/less than/equal to/not equal to)

TNC Specifications

Block execution time	4 ms/block TNC 407: 25 ms/block		
Control loop cycle time	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	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	TNC 425: max. 300 m/min (11 810 ipm) TNC 415 B: max. 30 m/min (1181 ipm) TNC 407: max. 30 m/min (1181 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: 1 µm Max. 99 999.999 mm (3937 in.) or 99 999.999°		

Accessories

FE 401 floppy disk unit

Description	Portable bench-top unit
Applications	All TNC contouring controls as well as TNC 131, TNC 135
Data interfaces	Two RS-232-C/V.24 interfaces
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
Diskettes	3.5" DS DD, 135 TPI

Triggering 3D touch probes

Description	Touch probe system with ruby tip and stylus with rated break point, standard shank for spindle insertion
Models	TS 120: Transmission via cable, integrated interface TS 511: Infrared transmission, separate transmitting and receiving units
Spindle insertion	TS 120: manual TS 511: automatic
Probing repeatability	Better than 1 µm (0.000 04 in.)
Probing speed	Max. 3 m/min (118 ipm)

Electronic handwheels

HR 130	• For panel mounting
HR 150	• Fixed-axis handwheel for the HRA 110 adapter
HR 330	• Portable version with cable transmission. Includes axis address keys, rapid traverse key, safety switch, emergency stop button.

11.6 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 containing 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 LBL number
- Note the input limits

EXT. IN-/OUTPUT NOT READY

Connect the external device properly.

FURTHER PROGRAM ENTRY IMPOSSIBLE

Erase some old files to make room for new ones.

JUMP TO LABEL 0 NOT PERMITTED

Do not program CALL LBL 0.

LABEL NUMBER ALLOCATED

A given label number can only be entered once in a 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 illegal 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.

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 switch off the control 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 ALLOCATED

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 511 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 COMPENSATION 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.

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.

11.7 Address Letters (ISO)

G functions

Group	G	Function
Positioning	00 01 02 03 05 06 07 10 11 12 13 15 16	Straight line interpolation, Cartesian coordinates, rapid traverse Straight line interpolation, Cartesian coordinates Circular interpolation, Cartesian coordinates, clockwise Circular interpolation, Cartesian coordinates, counterclockwise Circular interpolation, Cartesian coordinates, no direction of rotation given Circular interpolation, Cartesian coordinates, tangential contour transition Paraxial positioning block Straight line interpolation, polar coordinates, rapid traverse Straight line interpolation, polar coordinates Circular interpolation, polar coordinates, clockwise Circular interpolation, polar coordinates, counterclockwise Circular interpolation, polar coordinates, no direction of rotation given Circular interpolation, polar coordinates, tangential contour transition
Cycles	04 28 36 37 39 53 54 56 57 58 59 72 73 74 75 76 77 78 83 84 85 86	Dwell time Mirror image Oriented spindle stop Definition of the contour geometry Program call, cycle call with G79 Datum shift in datum table Datum shift in program Pilot drilling (in connection with G37) SLI Rough-out (in connection with G37) SLI Contour milling, clockwise (in connection with G37) SLI Contour milling, counterclockwise (in connection with G37) SLI Scaling factor Rotation of the coordinate system Slot milling Rectangular pocket milling, clockwise Rectangular pocket milling, counterclockwise Circular pocket milling, clockwise Circular pocket milling, counterclockwise Pecking Tapping with floating tap holder Rigid tapping Thread cutting
	120 121 122 123 124 125	Contour data Pilot drilling (in connection with G37) SLII Rough-out (in connection with G37) SLII Floor finishing (in connection with G37) SLII Side finishing (in connection with G37) SLII Contour train (in connection with G37)
	79	Cycle call
Select working plane	17 18 19 20	Working plane: XY, tool axis: Z Working plane: ZX, tool axis: Y Working plane: YZ, tool axis: X Tool axis: IV
Approach chamfer, rounding, depart contour	24 25 26 27	Chamfer with length R Corner rounding with R Tangential contour approach with R Tangential contour departure with R
	29	Transfer the last nominal position value as pole
Define blank form	30 31	Blank form definition for graphics, MIN point Blank form definition for graphics, MAX point
	38	Stop program run
Tool path compensation	40 41 42 43 44	No tool radius compensation (R0) Tool radius compensation, left of the contour (RL) Tool radius compensation, right of the contour (RR) Paraxial compensation, lengthening (R+) Paraxial compensation, shortening (R–)
	51 55	Next tool number (with central tool file) Probing function
Unit of measurement	70 71	Inches (at start of program) Millimeters (at start of program)
Dimensioning	90 91	Absolute dimensions Incremental dimensions
	98	Set label number
	99	Tool definition

Address letter	Function
%	Beginning of program or program call with G39
A B C	Rotary motion about the X axis Rotary motion about the Y axis Rotary motion about the Z axis
D	Parameter definition (program parameter Q)
F F F	Feed rate Dwell time with G04 Scaling factor with G72
G	Preparatory function
H H	Angle for polar coordinates in incremental/absolute dimensions Rotational angle with G73
I J K	X coordinate of circle center/pole Y coordinate of circle center/pole Z coordinate of circle center/pole
L L L	Set label number with G98 Go to label number Tool length with G99
M	Miscellaneous function
N	Block number
P P	Cycle parameter in fixed cycles Parameter in parameter definitions
Q	Program parameter/cycle parameter Q
R R R R R	Polar coordinate radius Circle radius with G02/G03/G05 Rounding radius with G25/G26/G27 Chamfer with G24 Tool radius with G99
S S	Spindle speed Oriented spindle stop with G36
T T	Tool definition with G99 Tool call
U V W	Linear motion parallel to the X axis Linear motion parallel to the Y axis Linear motion parallel to the Z axis
X Y Z	X axis Y axis Z axis
*	End of block

Parameter definitions

D	Function
00	Assignment
01 02 03 04	Addition Subtraction Multiplication Division
05	Square root
06 07	Sine Cosine
08	Root sum of squares ($c = \sqrt{a^2 + b^2}$)
09 10 11 12	If equal, jump If unequal, jump If larger, jump If smaller, jump
13	Angle (angle from $c \cdot \sin \eth$ and $c \cdot \cos \eth$)
14	Error number
15	Print
19	Assignment PLC marker

Sequence of Program Steps

Milling an outside corner

Program step	Key/ Function	Section in manual
1 Open or select program Entries: Program name Unit of measurement in program Blank form for graphic displays		4.4
2 Define tools Entries: Tool number Tool length Tool radius	G99	4.2
3 Call tool data Entries: Tool number Spindle axis Spindle speed	T	4.2
4 Tool change Entries: Coordinates of tool change position Radius compensation Feed rate (rapid traverse) Miscellaneous function (tool change)	G00	e.g. 5.4
5 Approach starting position Entries: Coordinates of starting position Radius compensation (G40) Feed rate (rapid traverse) Miscellaneous function (spindle ON clockwise)	G00/G40	5.2/5.4
6 Move tool axis to working depth	G00	
7 Approach contour Entries: Coordinates of first contour point Coordinate of (first) working depth Radius compensation for machining Machining feed rate	G01/G41/G42	5.2
8 Machining to last contour point Entries: Enter all required data for each contour element		5 to 8
9 Depart contour Entries: Coordinates of end position Feed rate (rapid traverse)	G00/G40	5.2
10 Retract Entries: Retract in the spindle axis Miscellaneous function (spindle stop, return)	G00 M02	
11 End of program		

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH
Dr.-Johannes-Heidenhain-Straße 5
D-83301 Traunreut, Deutschland
(08669) 31-0 56 831
FAX (08669) 50 6

t notice

Programming Guide

Contour cycles:

Sequence of program steps for machining with several tools

	⋮
List of subcontour programs	G37 P01 . . .
Drill – define/call Contour cycle: Pilot drilling Pre-position, cycle call	⋮ G56 P01 . . . ⋮
Roughing mill – define/call Contour cycle: Rough-out Pre-position, cycle call	⋮ G57 P01 . . . ⋮
Finishing mill – define/call Contour cycle: Contour milling Pre-position, cycle call	⋮ G58 P01 . . . ⋮
End of main program, return	**M02**
Contour subprograms	G98 ⋮ G98 L0

Radius compensation of the contour subprograms:

Contour	Sequence of programmed contour elements	Radius compensation
Inside (pocket)	Clockwise (CW) Counterclockwise (CCW)	G42 (RR) G41 (RL)
Outside (island)	Clockwise (CW) Counterclockwise (CCW)	G41 (RL) G42 (RR)

Coordinate transformations:

Coordinate transformation	Activate	Cancel
Datum shift	G54 X+20 Y+30 Z+10	G54 X+0 Y+0 Z+0
Mirror image	G28 X	G28
Rotation	G73 H+45	G73 H+0
Scaling factor	G72 F0,8	G72 F1

Q Parameter Definitions

D	Function	D	Function
00	Assign	**08**	Root sum of squares $c=\sqrt{a^2+b^2}$
01	Addition	**09**	If equal, go to label number
02	Subtraction	**10**	If not equal, go to label number
03	Multiplication	**11**	If greater than, go to label number
04	Division	**12**	If less than, go to label number
05	Square root	**13**	Angle from $c \sin \alpha$ and $c \cos \alpha$
06	Sine	**14**	Error number
07	Cosine	**15**	Print
		19	Assignment PLC

Addresses

Add.	Function
% **%**	Start of program Program call with G39
A **B** **C**	Rotary motion about X axis Rotary motion about Y axis Rotary motion about Z axis
D	Q parameter definitions
F **F** **F**	Feed rate Dwell time with G04 Scaling factor with G72
G	G functions
H **H**	Polar coordinate angle Angle of rotation with G73
I	X coordinate of the circle center/pole
J	Y coordinate of the circle center/pole
K	Z coordinate of the circle center/pole
L **L** **L**	Set a label number with G98 Go to a label number Tool length with G99
M	M functions

Add.	Function
N	Block number
P	Cycle parameter in fixed cycles
P	Value or Q parameter in Q parameter definition
Q	Q parameter
R **R** **R**	Polar coordinate radius Circle radius with G02/G03/G05 Rounding radius with G25/G26/G27
R	Tool radius with G99
S **S**	Spindle speed Oriented spindle stop with G36
T **T** **T**	Tool definition with G99 Tool call Next tool with G51
U **V** **W**	Axis parallel to X axis Axis parallel to Y axis Axis parallel to Z axis
X **Y** **Z**	X axis Y axis Z axis
*****	End of block

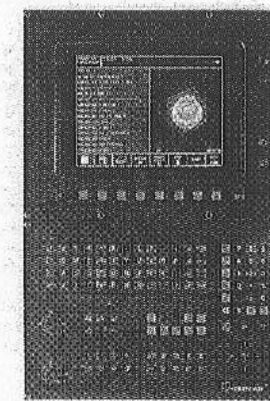
Program Example: Milling

Select the program number	PGM NAME
Program 234 in mm Define workpiece blank	% 234 G71 G30 G17 X+0 Y+0 Z–40 G31 G90 X+100 Y+100 Z+0
Tool definition Tool call Tool change position Tool call	G99 T1 L+0 R+5 T0 G17 G00 **G40** G90 Z+100 **M06** T1 G17 S1000
Starting position, next to the workpiece Working depth	X–20 Y–20 M03 Z–20
1st contour point, with radius compensation (RL) Tangential approach Straight line Chamfer Straight line Rounding Straight line Circle center Circle, incremental Last contour point, absolute	G01 **G41** X+0 Y+0 F200 G26 R15 Y+100 G24 R20 X+100 G25 R20 Y+25 I+100 J+0 G03 G91 X–25 Y–25 G01 G90 X+0 Y+0
Tangential departure End position, next to the workpiece Retract, return to start of program	G27 R15 G00 **G40** X–20 Y–20 Z+100 **M02**

HEIDENHAIN

TNC 407
TNC 415B
TNC 425

ISO Programming



Operating Modes

Machine/ programming — The keyboard and the display mode can be switched to "machine control" or programming" using the shift key on the visual display unit.

Machine control:

Manual — In this mode the axes can be moved with the machine axis direction buttons. Use the soft keys to enter the spindle speed, M functions and datum points, and to call the probing functions for the 3D touch probe.

Handwheel — Here the axes can be moved either with an electronic hand-wheel, or with the machine axis direction buttons after entering a jog increment (soft keys: see "Manual").

Positioning with MDI — This mode is for executing NC blocks which contain all information for a positioning move or machining step (also applies to feed rates, circle centers and cycles). The blocks are stored in the program $MDI.

Program run/ full sequence — When the program has been started with the machine START button, it runs automatically to its end or until it encounters a program STOP. The machining process can be observed on the screen with the **simultaneous graphics** feature (except TNC 407).

Program run/ single block — Each block must be started separately with the machine START button. The machining process can be observed on the screen with the **simultaneous graphics** feature (except TNC 407).

Programming:

Programming and editing — This mode allows you to edit HEIDENHAIN conversational and ISO programs, tool tables, datum tables, pallet tables and text files, and then downloaded or output them over the RS-232-C or RS-422 data interfaces.

Test program — The **test graphics** feature allows you to check part programs for errors before actual machining.

G Functions

Tool movement

G00 Straight line interpolation, Cartesian coordinates, rapid traverse
G01 Straight line interpolation, Cartesian coordinates
G02 Circular interpolation, Cartesian coordinates, clockwise
G03 Circular interpolation, Cartesian coordinates, counterclockwise
G05 Circular interpolation, Cartesian coordinates, no direction of rotation
G06 Circular interpolation, Cartesian coordinates, tangential contour transition
* G07 Paraxial positioning block
G10 Straight line interpolation, polar coordinates, rapid traverse
G11 Straight line interpolation, polar coordinates
G12 Circular Interpolation, polar coordinates, clockwise
G13 Circular interpolation, polar coordinates, counterclockwise
G15 Circular interpolation, polar coordinates, no direction of rotation
G16 Circular interpolation, polar coordinates, tangential contour transition

Chamfer/Rounding/Approach contour/Depart contour

* G24 Chamfer with length R
* G25 Corner rounding with radius R
* G26 Tangential contour approach with radius R
* G27 Tangential contour departure with radius R

Tool definition

* G99 With tool number T, length L, radius R

Tool radius compensation

G40 No tool radius compensation
G41 Tool radius compensation, left of the contour
G42 Tool radius compensation, right of the contour
G43 Paraxial compensation for G07, lengthening
G44 Paraxial compensation for G07, shortening

Blank for definition for graphics

G30 (G17/G18/G19) MIN point
G31 (G90/G91) MAX point

Simple fixed cycles

G83 Pecking
G84 Tapping with floating tap holder
G85 Rigid tapping
G86 Thread cutting
G74 Slot milling
G75 Rectangular pocket milling, clockwise
G76 Rectangular pocket milling, counterclockwise
G77 Circular pocket milling, clockwise
G78 Circular pocket milling, counterclockwise

SL cycles, group 1

G37 Contour geometry, list of subcontour program numbers
G56 Pilot drilling
G57 Rough-out
G58 Contour milling, clockwise (finishing)
G59 Contour milling, counterclockwise (finishing)

* Non-modal function

G Functions

SL cycles, group 2

G37 Contour geometry, list of subcontour program numbers
G120 Contour data (applies to G121 to G124)
G121 Pilot drilling
G122 Rough-out
G123 Floor finishing
G124 Side finishing
G125 Contour train (machine open contour)

Coordinate transformations

G53 Datum shift in datum table
G54 Datum shift in program
G28 Mirror image
G73 Rotation of the coordinate system
G72 Scaling factor (reduce or enlarge contour)

Special cycles

* G04 Dwell time F (in seconds)
G36 Oriented spindle stop
* G39 Program call

Define working plane

G17 Working plane: X/Y; tool axis: Z
G18 Working plane: Z/X; tool axis: Y
G19 Working plane: Y/Z; tool axis: X
G20 Tool axis: IV

Dimensioning

G90 Absolute dimensions
G91 Incremental dimensions

Unit of measurement

G70 Inches (define at start of program)
G71 Millimeters (define at start of program)

Other G functions

G29 Transfer the last nominal position value as a pole (circle center)
G38 Stop program run
* G51 Next tool number (with central tool file)
G55 Probing function
* G79 Cycle call
* G98 Set label number

* Non-modal function

M Functions

M00	Stop program run/Spindle stop/Coolant off
M02	Stop program run/Spindle stop/Coolant off delete status display (depending on machine parameter) Return to block 1
M03 M04 M05	Spindle ON clockwise Spindle ON counterclockwise Spindle stop
M06	Tool change/spindle stop (depending on machine parameter)/ Stop program run
M08 M09	Coolant ON Coolant OFF
M13 M14	Spindle ON clockwise/Coolant ON Spindle ON counterclockwise/Coolant ON
M30	Same as M02
M89 M99	Vacant miscellaneous function or Cycle call, modal Cycle call, non-modal
M90	Constant contouring speed at inside corners and uncompensated corners
M91 M92	Coordinates in positioning block are referenced to the machine datum Coordinates in positioning block are referenced to a position defined by the machine builder
M93 M94 M95 M96	Reserved Reduce display of rotary axis to value under 360° Reserved Reserved
M97 M98	Path compensation on outside corners: points of intersection instead of transition arc End of path compensation, non-modal
M101 M102	Automatic tool change with sister tool if maximum tool life has expired Reset M101
M103 M104	Reduce plunging rate to factor F (percent) Reserved
M105 M106	Machining with first Kv factor Machining with second Kv factor
M107 M108	Suppress error message with sister tools with oversize (with blockwise transfer) Reset M107
M109 M110 M111	Constant contouring speed at the tool cutting edge on inside and outside corners Constant contouring speed at the tool cutting edge on inside corners Feed rate refers to the tool path center (standard setting)
M112 M113	Insert rounding arc between two straight lines, enter tolerance E Reset M112
M114 M115	Automatic compensation of the machine geometry when working with swivel axes Reset M114