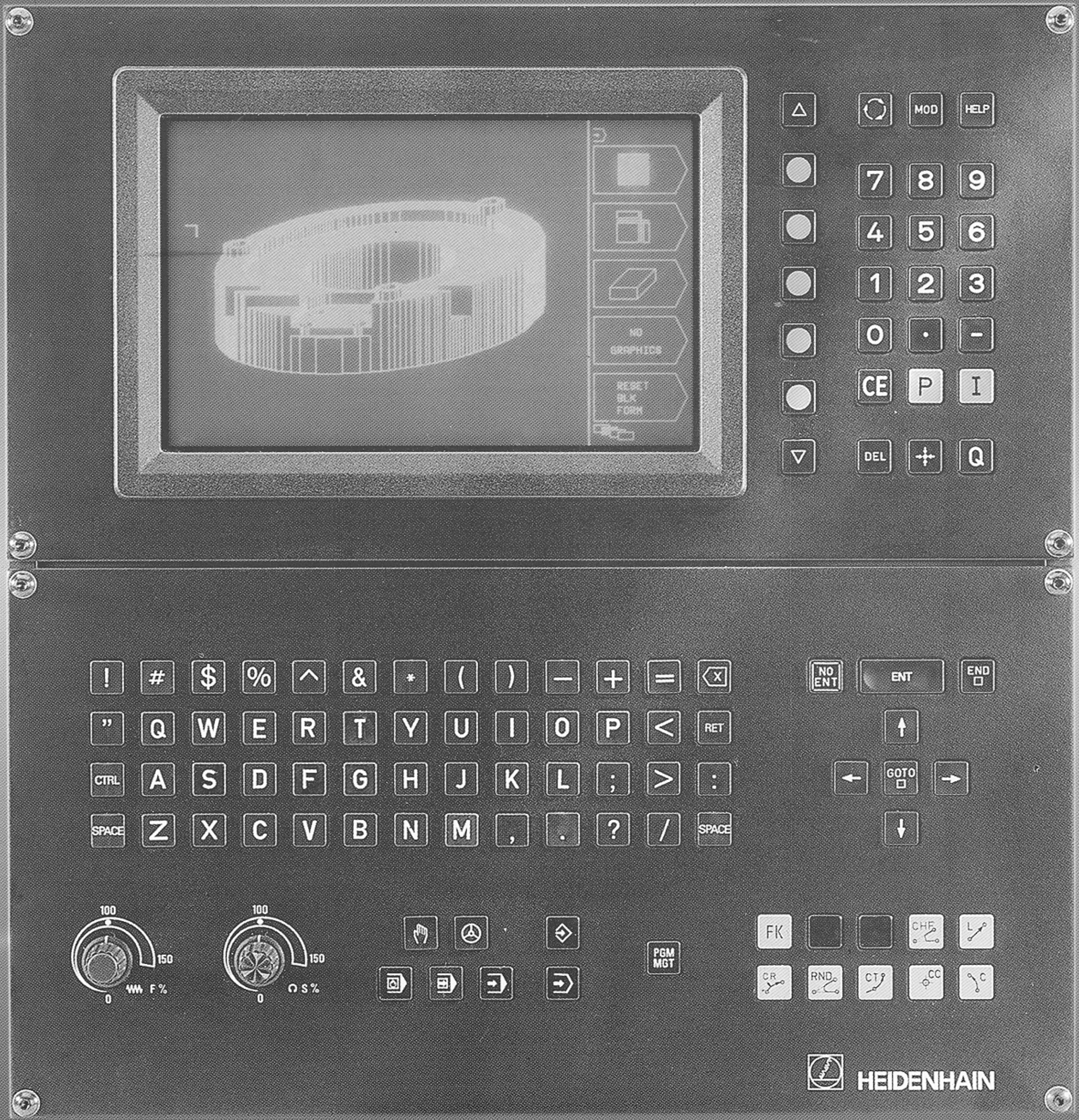


HEIDENHAIN

User's Manual
ISO Programming

TNC 370

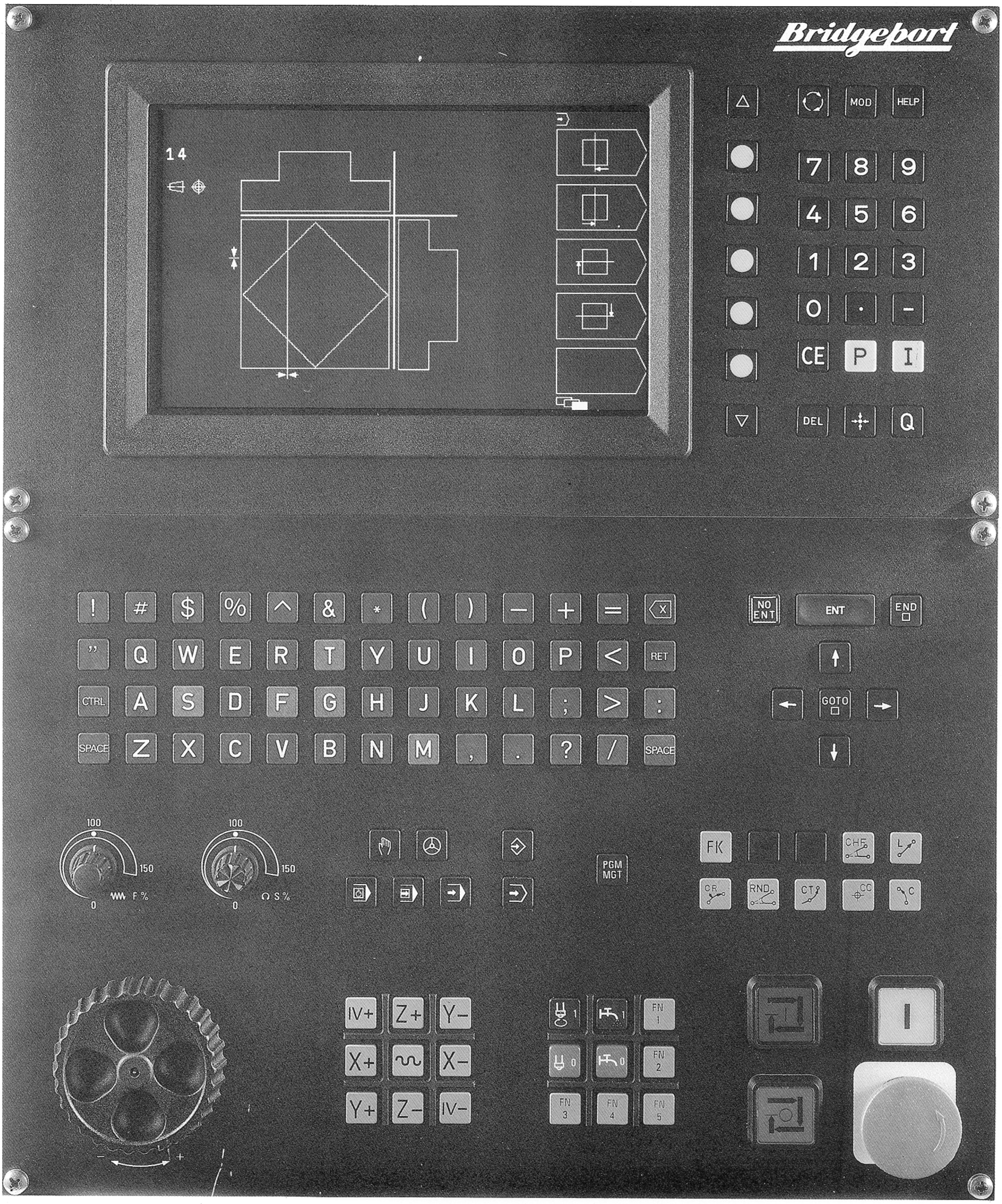
February 1996

TNC Guideline:

From the workpiece drawing to program-controlled machining

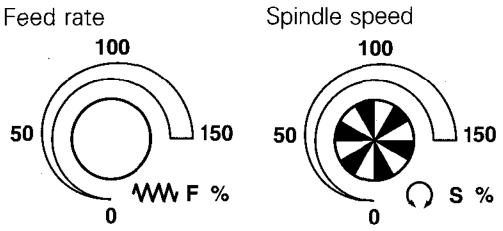
Step	Task	TNC operat. mode	Refer to section
	Preparation		
1	Select tools	——	——
2	Set workpiece datum for coordinate system	——	——
3	Determine spindle speeds and feed rates	——	——
4	Switch on machine	——	1.3
5	Cross over reference marks	✋ or ⎈	1.3, 2.1
6	Clamp workpiece	——	——
7	Set the datum / reset position display ...		
7a	... **with** a 3D touch probe	✋ or ⎈	2.5
7b	... **without** a 3D touch probe	✋ or ⎈	2.3
	Entering and testing part programs		
8	Enter part program or download over external data interface	⟐ or PGM MGT	5 to 8, 9
9	Test program for errors	⇥	3.1
10	Test run: Run program block by block without tool	⟐	3.2
11	If necessary: Optimize the program	⟐	5 to 8
	Machining the workpiece		
12	Insert tool and run part program	⇥	3.2

TNC 370 with TE 370 Keyboard Unit



Keys and Controls on the TNC 370

Override knobs



Machine keys *(only available on TE 370 keyboard unit)*

I	0	NC start/NC stop
1	0	Coolant on/off
1	0	Spindle on/off
FN 1 ...	FN 5	Machine functions (see machine operating manual)
X+ ...	Z+	Axis direction keys

Visual display unit

●		Soft keys: for selecting function on screen
▽	△	Shift the soft keys
○		Select screen layout

Typewriter keyboard for entering letters and symbols

Q W E R T Y	File names/ Comments
G F S T M	ISO programs

Machine operating modes

	MANUAL OPERATION
	ELECTRONIC HANDWHEEL
	POSITIONING WITH MANUAL DATA INPUT
	PROGRAM RUN, SINGLE BLOCK
	PROGRAM RUN, FULL SEQUENCE
	PROGRAMMING AND EDITING
	TEST RUN

Cursor keys and GOTO

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

Program management

PGM MGT	For selecting, protecting, renaming, erasing, inputting and outputting programs

Programmable contours *(conversational programming only)*

FK	FK Free contour programming
CHF	Chamfer
L	Straight line
CC	Circle center / Pole for polar coordinates
C	Circle around center point/pole
CR	Circle with radius
CT	Circle with tangential connection
RND	Corner rounding

Numerical entry, editing

0 ...	9	Numerical values
•	−	Decimal point, algebraic sign
I	P	Incremental values, polar coordinates
Q		Q parameters for part families or in mathematical functions
CE		Clear numerical entry, or TNC message
✛		Actual position capture
ENT		Confirm entry and resume dialog
NO ENT		Ignore dialog prompts and delete words
DEL		Abort dialog, delete program sections
END □		Conclude block
MOD		MOD functions
HELP		HELP function

How to use this manual

This manual describes the functions and features available on the TNC 370 with NC software number 280 600 07 or higher.

This manual describes all available functions of the TNC 370. For information on your machine tool, refer to the machine tool manual.

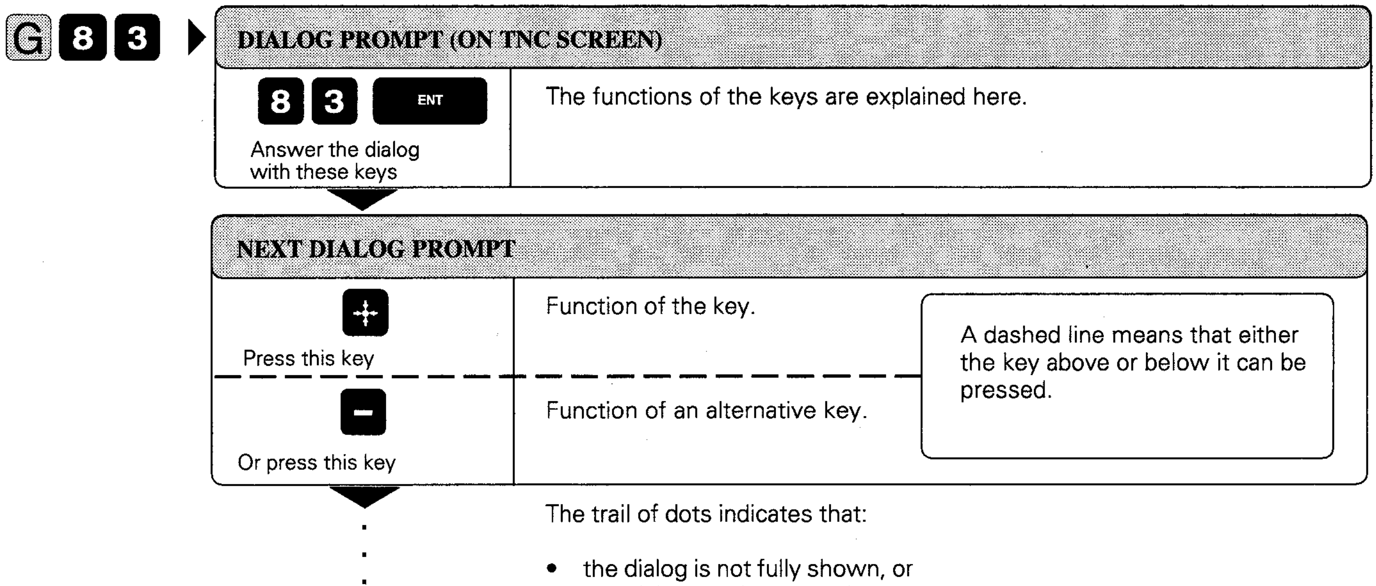
If you're a beginner with NC, you can use this manual as a step-by-step workbook. The first part of the manual deals with the basics of NC technology and describes the TNC functions. It then introduces the technique of conversational programming. Each new function is thoroughly described when it is first introduced, and the numerous examples can be tried out directly on the control — so you won't get lost in the theory. You should work through this manual 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 NC, you can use this manual as a comprehensive review and reference guide. The table of contents and cross references help you quickly find the topics and information you need. Easy-to-read dialog flowcharts show you how to enter the data required for each function.

The dialog flowcharts consist of sequentially arranged instruction boxes. Each key is illustrated next to an explanation of its function, to aid you when performing the operation for the first time. The experienced user can use the key sequences illustrated in the left part of the flowchart as a quick overview. The TNC dialogs are always presented on a gray background.

Layout of the dialog flowcharts

Dialog initiation key



The functions of the keys are explained here.

Function of the key.

Function of an alternative key.

A dashed line means that either the key above or below it can be pressed.

The trail of dots indicates that:

- the dialog is not fully shown, or
- the dialog continues on the next page.

]

Contents User's Manual ISO Programming TNC 370

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, HELP Function

11 Tables and Overviews

1 Introduction

1.1 The TNC 370

Control

The TNC 370 is a shop-floor programmable contouring control for milling machines with up to four axes. The spindle can be rotated to a specified angular stop position (oriented spindle stop).

Visual display unit and operating panel

The flat luminescent monochrome screen clearly displays all information necessary for operating the TNC 370. The keys on the operating panel are grouped according to their functions. This simplifies operating and programming the control. The following keyboard units are available:

- TE 370 with machine operating keys
  (such as NC start, NC stop, coolant on/off)
- TE 371 without machine operating keys

Programming

The TNC is programmed according to ISO. It can also be programmed in easy-to-use HEIDENHAIN conversational format, which is described in a separate User's Manual.

Graphics

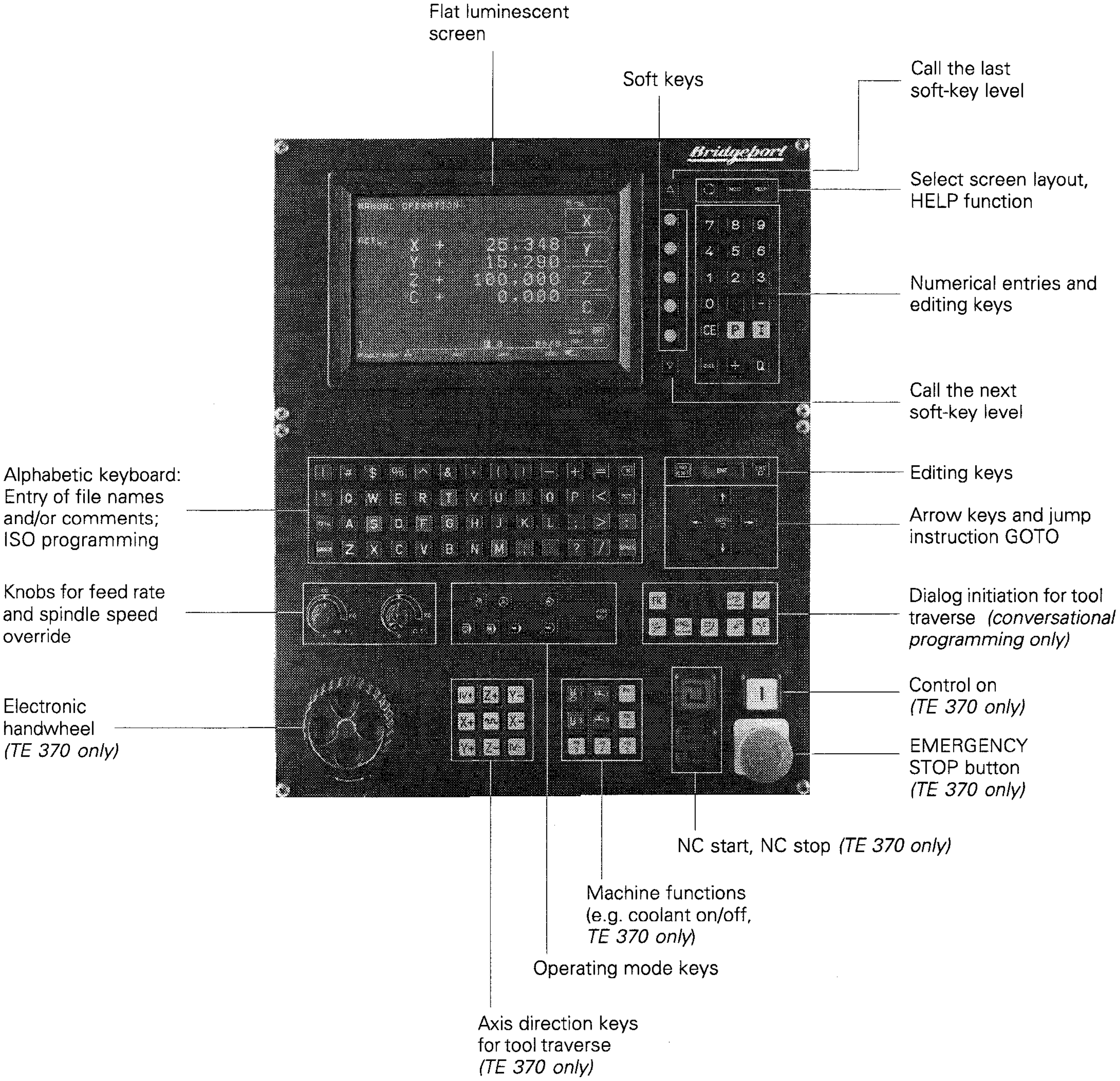
The graphic simulation feature enables you to test programs before actual machining. Several types of graphic representation can be selected.

Compatibility

The TNC 370 can execute any part program that was created on a HEIDENHAIN control model TNC 150B or later.

The operating panel with the TE 370 keyboard unit

The flat luminescent screen, control keys, and all machine operating keys are integrated in the operating panel. If you are using the TE 371 keyboard unit, the machine operating keys are not available.



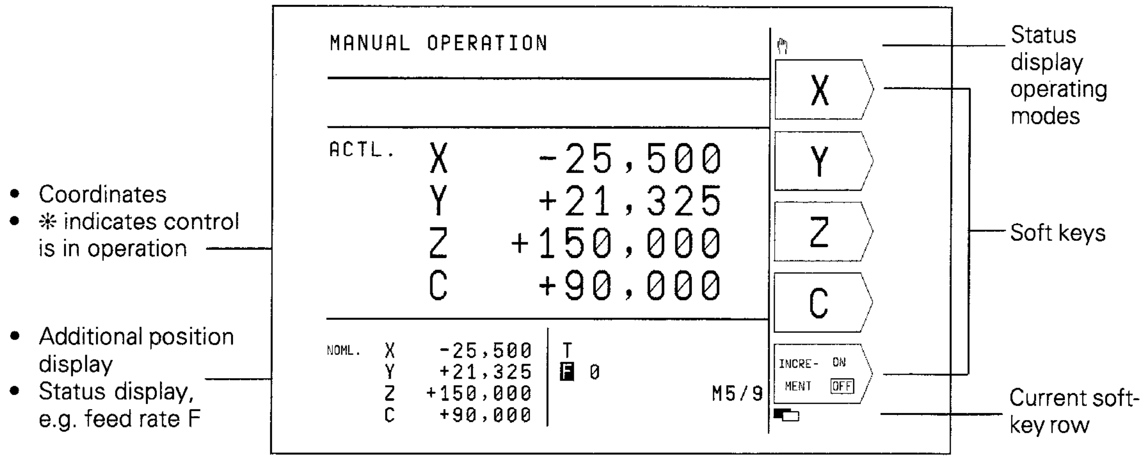
Screen layout

Use the ○ key to select the screen layout.
These are the available layouts:

Modes of operation	Screen contents	Soft key
PROGRAM RUN / FULL SEQUENCE PROGRAM RUN / SINGLE BLOCK TEST RUN	Program text only	TEXT
	Left: Program text Right: Program information	TEXT PGM STATUS
	Left: Program text Right: Additional position display	TEXT POS. STATUS
	Left: Program text Right: Tool information	TEXT TOOL STATUS
	Left: Program text Right: Active coordinate transformations	TEXT COORD.TRANS. STATUS
	(Not in the TEST RUN mode of operation) Left: Program text Right: Results of automatic tool measurement	TEXT TOOL PROBE STATUS
PROGRAMMING AND EDITING	Program text only	TEXT
	Programming graphics	GRAPHICS
	Left: Program text Right: Programming graphics	TEXT GRAPHICS
	Left: Program text Right: Graphics only during cycle programming	TEXT FIGURE

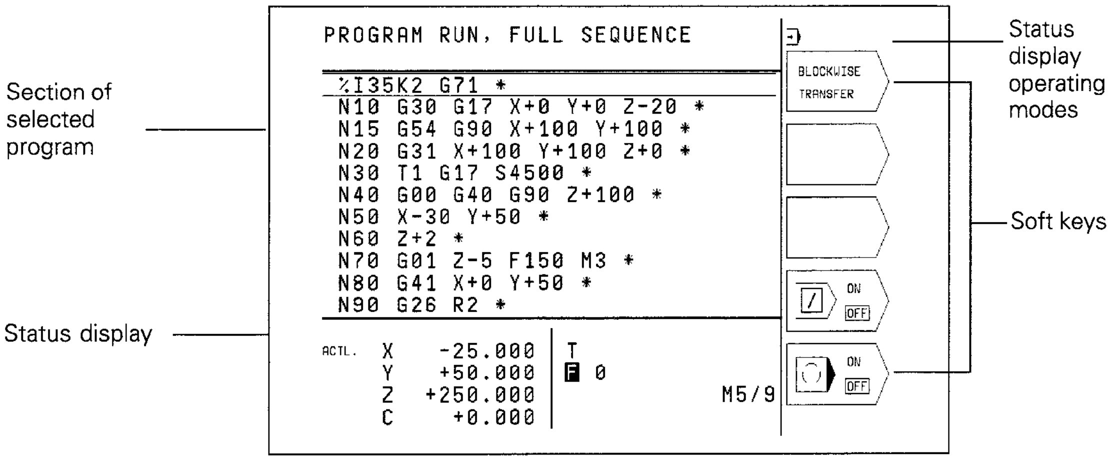
The screen layout cannot be changed in the operating modes MANUAL, HANDWHEEL, and POSITIONING WITH MANUAL DATA INPUT.

MANUAL OPERATION, EL. HANDWHEEL and
POSITIONING WITH MANUAL DATA INPUT modes of operation:



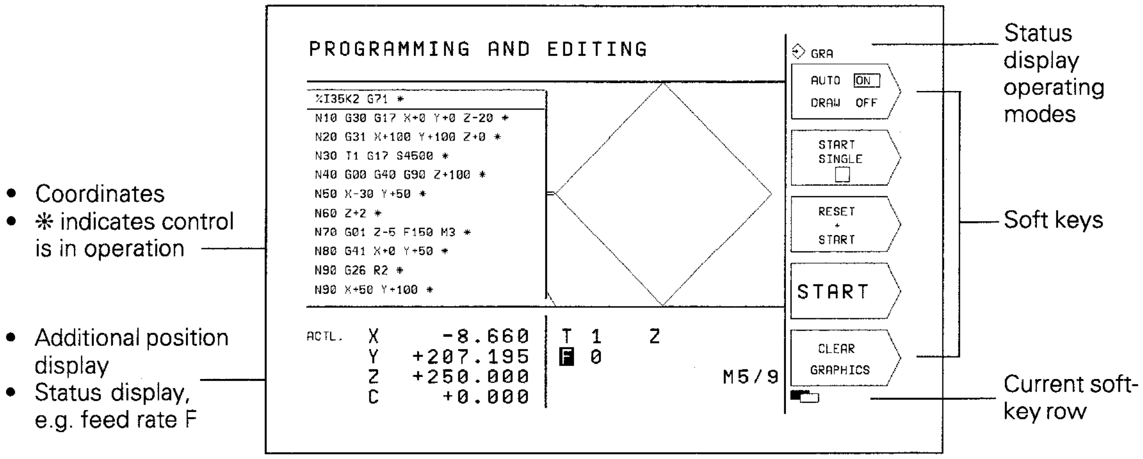
```
MANUAL OPERATION

ACTL.  X     -25,500
       Y     +21,325
       Z    +150,000
       C     +90,000

NOML.  X     -25,500 | T
       Y     +21,325 | F 0
       Z    +150,000 |          M5/9
       C     +90,000 |
```

PROGRAM RUN FULL SEQUENCE and
PROGRAM RUN SINGLE BLOCK modes of operation (TEXT):



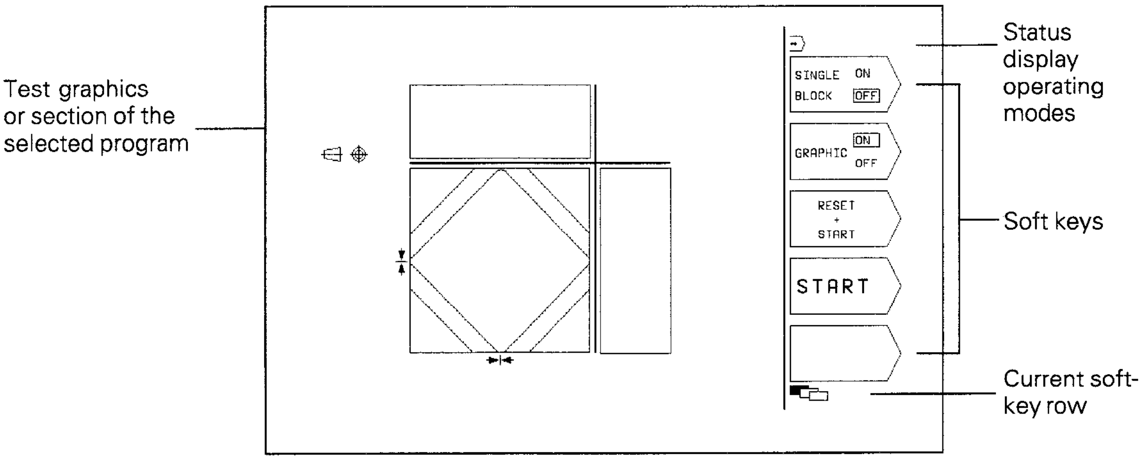
```
PROGRAM RUN, FULL SEQUENCE

%I35K2 G71 *
 N10 G30 G17 X+0 Y+0 Z-20 *
 N15 G54 G90 X+100 Y+100 *
 N20 G31 X+100 Y+100 Z+0 *
 N30 T1 G17 S4500 *
 N40 G00 G40 G90 Z+100 *
 N50 X-30 Y+50 *
 N60 Z+2 *
 N70 G01 Z-5 F150 M3 *
 N80 G41 X+0 Y+50 *
 N90 G26 R2 *

ACTL.  X    -25.000 | T
       Y    +50.000 | F 0
       Z   +250.000 |           M5/9
       C     +0.000 |
```

PROGRAMMING AND EDITING mode of operation (TEXT+GRAPHICS):



```
PROGRAMMING AND EDITING

%I35K2 G71 *
N10 G30 G17 X+0 Y+0 Z-20 *
N20 G31 X+100 Y+100 Z+0 *
N30 T1 G17 S4500 *
N40 G00 G40 G90 Z+100 *
N50 X-30 Y+50 *
N60 Z+2 *
N70 G01 Z-5 F150 M3 *
N80 G41 X+0 Y+50 *
N90 G26 R2 *
N90 X+50 Y+100 *

ACTL.  X      -8.660 | T 1    Z
       Y    +207.195 | F 0
       Z    +250.000 |             M5/9
       C      +0.000 |
```

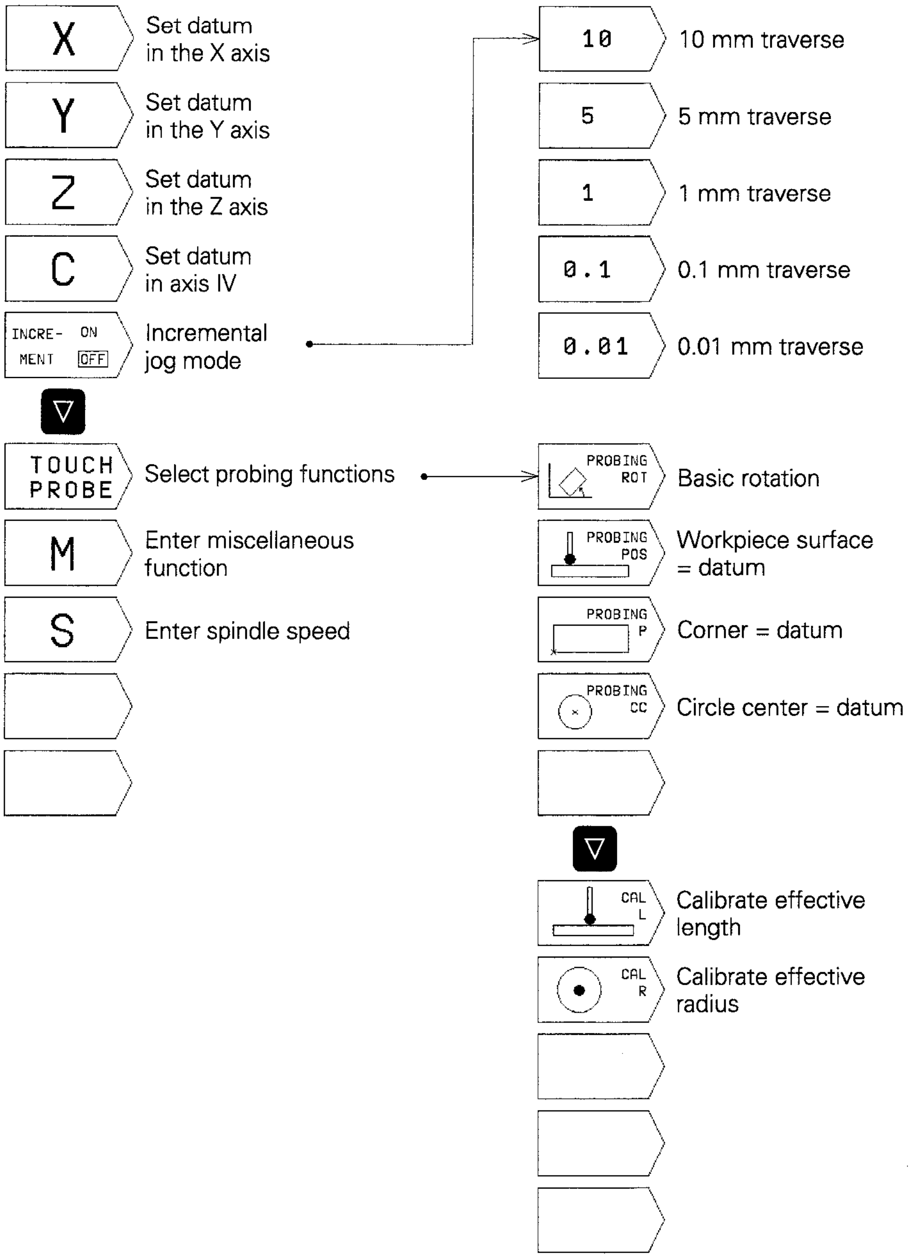
TEST RUN operating mode (GRAPHICS):



Soft key structure

Soft keys are used to activate and program most functions. A soft key is a key whose function is controlled by the software. The individual soft key functions are arranged in a menu tree structure. This allows multiple functions to be assigned to one key. The following section shows the menu tree of the TNC 370 and will help you to select a specific function, particularly in the beginning.

MANUAL mode of operation



ELECTRONIC HANDWHEEL mode of operation

X	Set datum in X axis with handwheel
Y	Set datum in Y axis with handwheel
Z	Set datum in Z axis with handwheel
C	Set datum in axis IV with handwheel
INCRE- MENT ON OFF	Incremental jog mode
▽	
TOUCH PROBE	Select probing functions
M	Enter miscellaneous function
S	Enter spindle speed

10	10 mm traverse
5	5 mm traverse
1	1 mm traverse
0.1	0.1 mm traverse
0.01	0.01 mm traverse

PROBING ROT	Basic rotation
PROBING POS	Workpiece surface = datum
PROBING P	Corner = datum
PROBING CC	Circle center = datum
▽	
CAL L	Calibrate effective length
CAL R	Calibrate effective radius

PROGRAM RUN FULL SEQUENCE and PROGRAM RUN SINGLE BLOCK modes of operation

BLOCKWISE TRANSFER	Blockwise transfer
/ ON OFF	Block skip active/inactive
0 ON OFF	Programmed stop (M01) active/inactive

PROGRAMMING AND EDITING mode of operation

AUTO ON DRAW OFF	Automatic drawing on/off
START SINGLE ☐	Start graphics blockwise
RESET + START	Start graphics from beginning of program
START	Start graphics from current block
CLEAR GRAPHICS	Reset graphics
▽	
≫	Enlarge zoom range
≪	Reduce zoom range
WINDOW DETAIL	Show selected detail
WINDOW BLK FORM	Show defined blank form

PROGRAMMING AND EDITING mode of operation, Program Management

PGM MGT

EXT	Data input/output →
PROTECT/ UNPROTECT	Protect program
RENAME	Rename program
CLEAR PGM	Erase program ↓
YES	Confirm erasure
NO	Abort erasure

TRANSFER TNC ← EXT	Input all programs
TRANSFER TNC ?← EXT	Input offered program
TRANSFER TNC ← EXT	Input selected program
TRANSFER TNC → EXT	Output selected program
TRANSFER TNC → EXT	Output all programs
▽	
SHOW EXT DIRECTORY	Show programs in external storage

TEST RUN mode of operation

In the Test Run mode of operation the soft-key structure varies according to the selected display mode. The TNC provides three sets of soft-key assignments:

SINGLE BLOCK ON OFF	Test run in Full Sequence/ Single block
GRAPHIC ON OFF	Test run with/without graphics
RESET + START	Start test run from beginning of program
START	Start test run from current block
STOP AT N	Test run up to block number N

▽

Plan view or **fast internal image generation** is active:

■	Plan view
⊞	Projection in three planes
▱	3D view
NO GRAPHICS	Fast internal image generation
RESET BLK FORM	Reset blank form

Projection in three planes is active:

■	Plan view
⊞	Projection in three planes
▱	3D view
NO GRAPHICS	Fast internal image generation
RESET BLK FORM	Reset blank form

▽

→	Move sectional plane to the right
←	Move sectional plane to the left
↑	Move sectional plane upward
↓	Move sectional plane downward

3D view is active:

■	Plan view
⊞	Projection in three planes
▱	3D view
NO GRAPHICS	Fast internal image generation
RESET BLK FORM	Reset blank form

▽

↻	Rotate 3D graphic (90° in pos. direction)
↺	Rotate 3D graphic (90° in neg. direction)

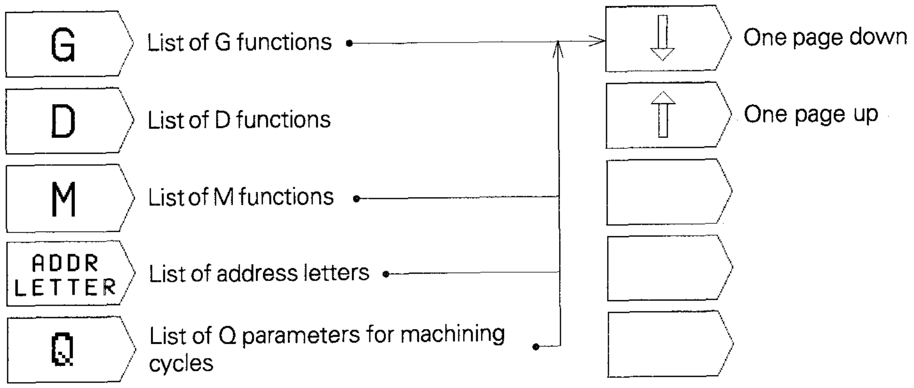
▽

⊡	Zoom function
+	Move sectional plane in positive direction
−	Move sectional plane in negative direction
TRANSFER DETAIL	Transfer detail
WINDOW BLK FORM	Show defined blank form

MOD function

USER PARAM.	User parameters
AXIS LIMITS	Set axis traverse limits
RS 232 SETUP	Set data interface
INFO SYSTEM	Show NC- and PLC-software numbers, free memory
🔑	Enter code number

HELP function



TNC accessories

3D touch probe systems

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

- Automatic workpiece alignment (i.e. compensation of workpiece misalignment)
- Datum setting
- Measurement of the workpiece during program run
- Digitizing 3D surfaces (option, only available with conversational programming)
- Tool measurement with the TT 120 touch probe (only available with conversational programming)

The TS 220 transmits signals via cable, the TS 630 via infrared light.

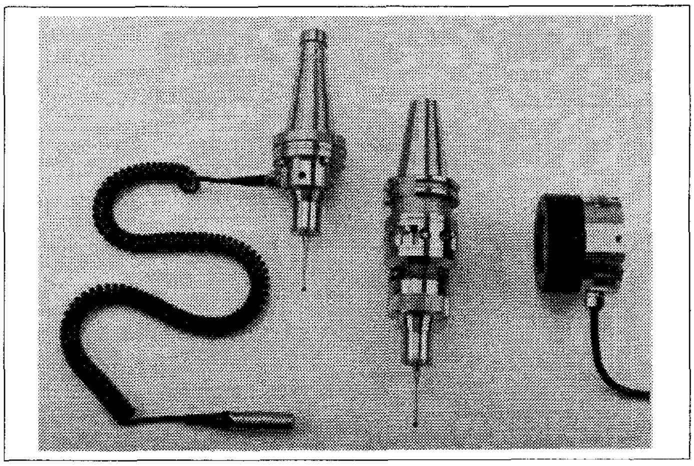


Fig. 1.6: HEIDENHAIN 3D touch probes TS 220 and TS 630

Floppy disk unit

The HEIDENHAIN FE 401 floppy disk unit serves as an external memory for the TNC, enabling you to store programs and tables on diskette.

With the FE 401 you can transfer programs that were written on a PC to the TNC. Very large programs that exceed the storage capacity of the TNC can be "drip fed" block-by-block: The machine executes the transferred blocks and erases them immediately, freeing up memory for further blocks from the FE.

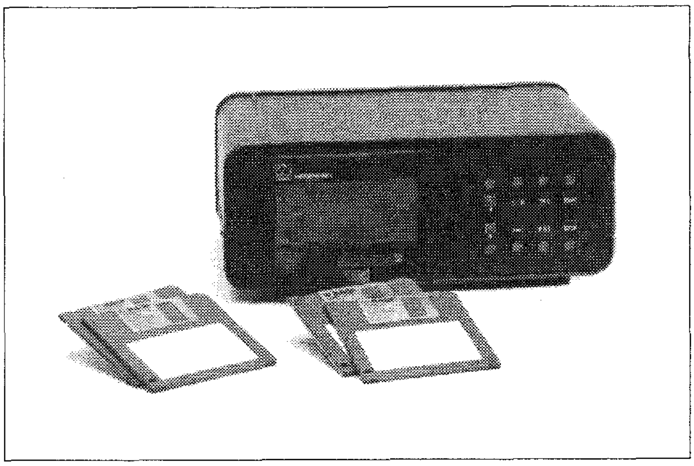


Fig. 1.7: HEIDENHAIN FE 401 floppy disk unit

Electronic handwheel

An electronic handwheel gives you 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. The traverse distance per handwheel revolution can be adjusted over a wide range.

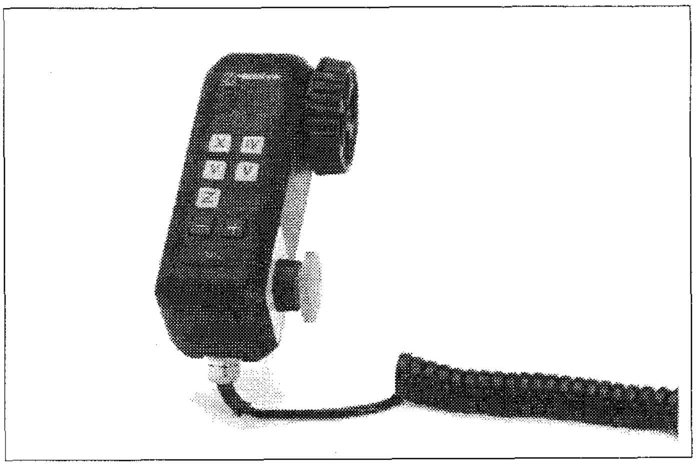


Fig. 1.8: HEIDENHAIN HR 330 electronic handwheel

1.2 Fundamentals of NC

Introduction

This chapter addresses the following topics:

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

What is NC?

NC stands for Numerical Control. Simply put, numerical control is the operation of a machine by means of a series of coded instructions. A modern control such as the TNC 370 has a built-in computer for this purpose and is therefore also called CNC (Computerized Numerical Control).

The part program

A part program is a complete list of instructions for machining a workpiece. It contains such information as the target position of a tool movement, the tool path (how the tool is to move towards the target position) and the feed rate. The program must also contain information on the radius and length of the tools, the spindle speed and the tool axis.

Programming

Program input according to ISO is partially dialog prompted. Except for G90/G91, the individual commands, called *words*, can be entered in any order within a block. The TNC automatically sorts the commands when you have completed the block.

The TNC can also be programmed in plain-language conversational dialog or in DNC mode.

Reference system

A reference system is needed in order to define positions. 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 longitude and latitude lines around the globe constitutes an absolute reference system — in contrast to the relative definition of a position that is referenced to another known location.

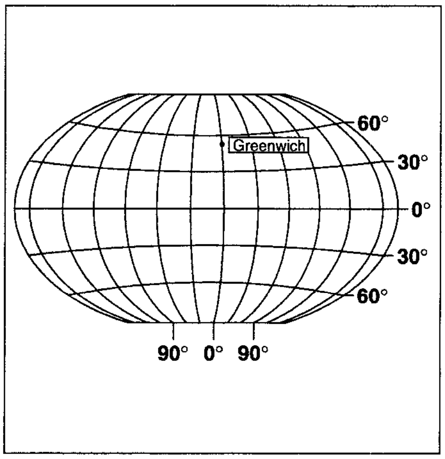


Fig. 1.9: The geographic coordinate system is an absolute reference system

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 René Descartes, who lived from 1596 to 1650). The Cartesian coordinate system is based on three coordinate axes X, Y and Z that 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.

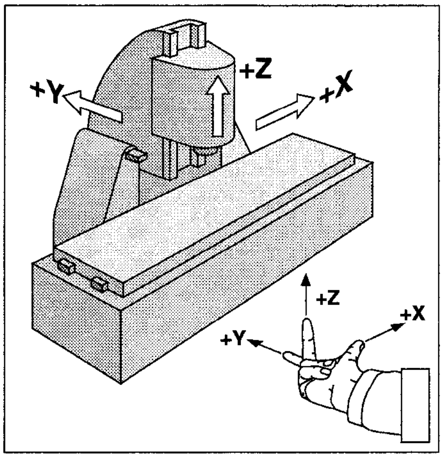


Fig. 1.10: Designations and directions of the axes on a milling machine

Additional axes

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

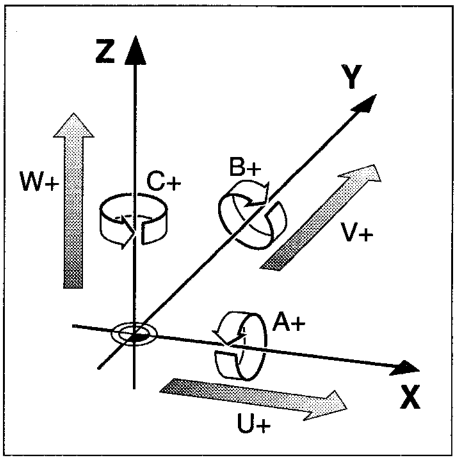


Fig. 1.11: Directions and designations of the additional axes

Polar coordinates

The Cartesian coordinate system is especially useful for parts whose dimensions are mutually perpendicular. But when workpieces contain circular arcs, or when the dimensions are given in degrees, it is often easier to use polar coordinates. While Cartesian coordinates are three-dimensional and can describe points in space, polar coordinates are two dimensional and can only 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 that pole and the angle of the line from 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.

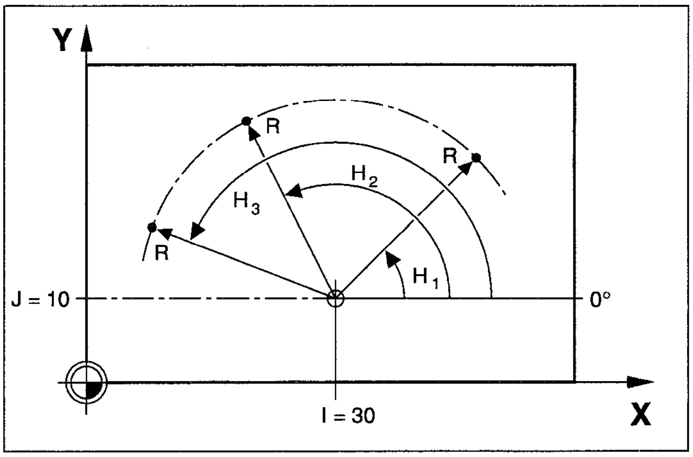


Fig. 1.12: Positions on an arc with polar coordinates

Positions in this plane are defined by:

- **Polar Radius R** – The distance from pole I,J to the defined position.
- **Polar Angle H** – The angle between the reference axis and the scale.

Setting a pole at I, J, K

The pole is defined by setting two Cartesian coordinates. These two 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

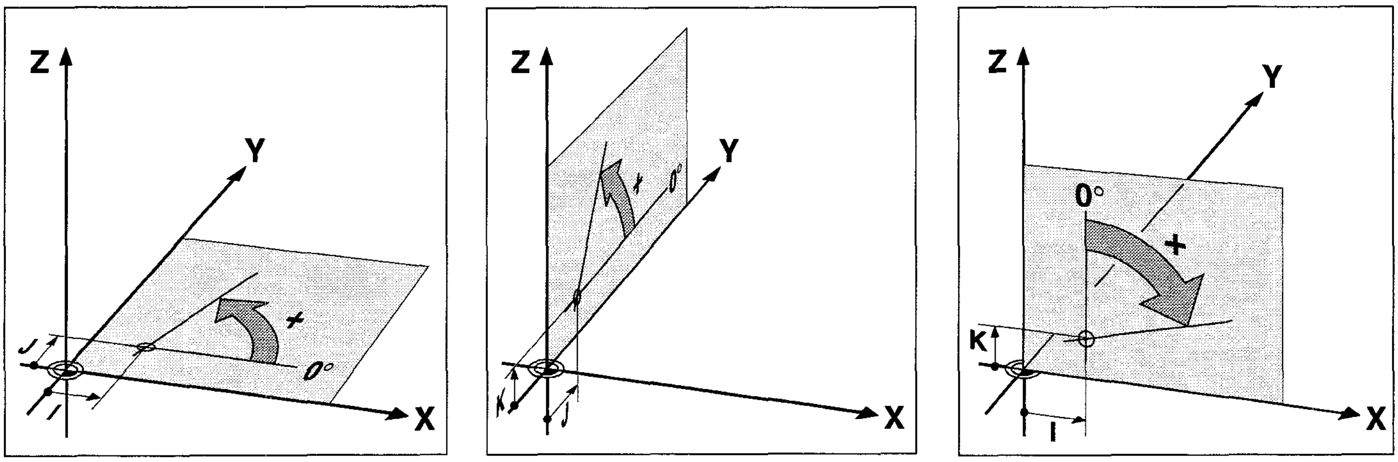


Fig. 1.13: Polar coordinates and their associated reference axes

Setting the datum

The workpiece drawing identifies a certain prominent point on the workpiece (usually a corner) as the absolute datum, and perhaps one or more other points as relative datums. The datum setting process 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 (such as to compensate the tool radius).

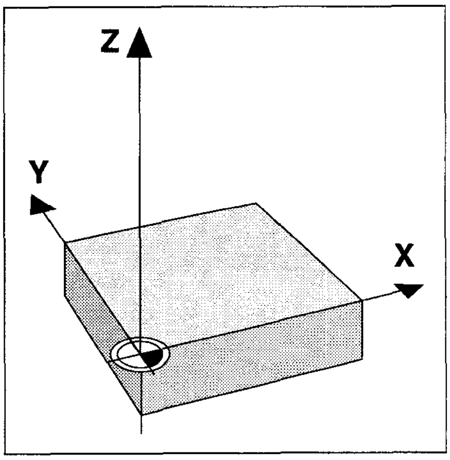
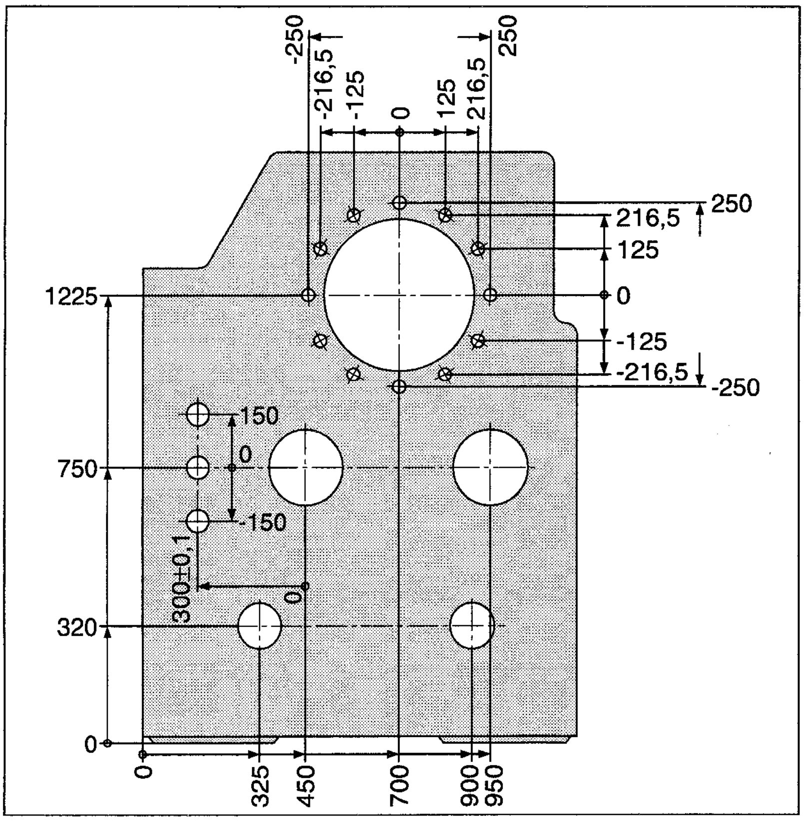


Fig. 1.14: The workpiece datum serves as the origin of the Cartesian coordinate system

Example:

Drawings with several relative datums (according to ISO 129 or DIN 406, Part 11; Figure 171)



Example:

Coordinates of point ①:

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

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

A 3D touch probe from HEIDENHAIN is an especially convenient and efficient way to find and set datums.

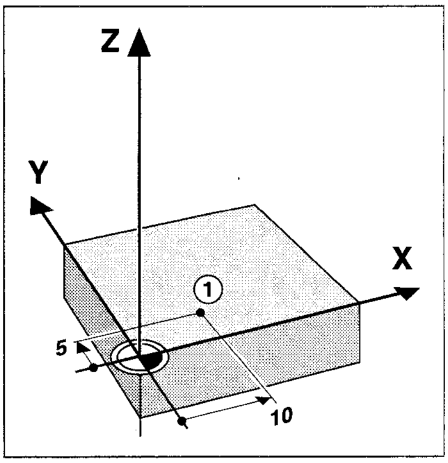


Fig. 1.16: Point ① defines the coordinate system.

Absolute workpiece positions

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

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

When you drill or mill a workpiece according to a workpiece drawing with absolute coordinates, you move the tool **to** the coordinates. Absolute coordinates are identified with G90.

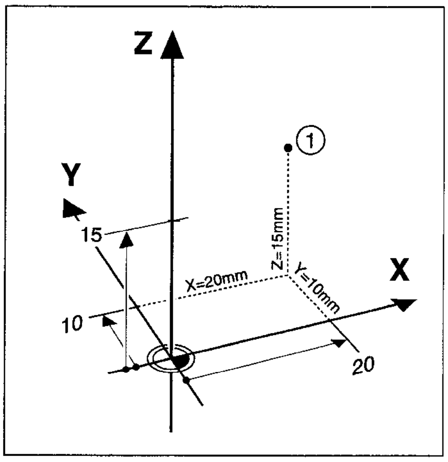


Fig. 1.17: Position definition through absolute coordinates

Incremental workpiece positions

A position can be referenced to the previous nominal position. That is, the relative datum is always the last programmed position. Such coordinates are referred to as **incremental coordinates** (increment = growth). They are also called incremental or chain dimensions (since the positions are defined as a chain of dimensions). Incremental coordinates are identified with G91.

Example: Incremental coordinates of the position ③
referenced to position ②

Absolute coordinates of the position ② :
X = 10 mm
Y = 5 mm
Z = 20 mm

Incremental coordinates of the position ③ :
IX = 10 mm
IY = 10 mm
IZ = –15 mm

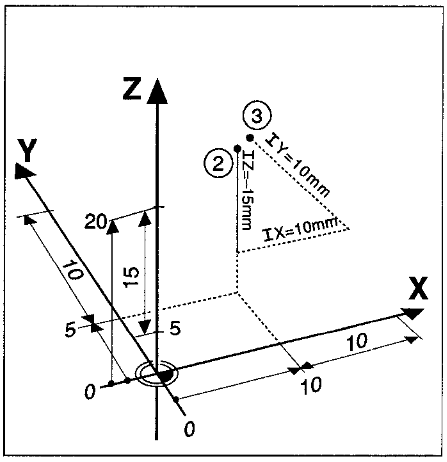


Fig. 1.18: Position definition through incremental coordinates

When you drill or mill a workpiece according to a workpiece drawing with incremental coordinates, you move the tool **by** the coordinates.

An incremental position definition is therefore an immediately relative definition. This is also the case when a position is defined by the **distance-to-go** to the target position (in this case the relative datum is located at the target position). The distance-to-go has a negative algebraic sign if the target position lies in the negative axis direction from the actual position.

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

- **Absolute polar coordinates** always refer to the pole I,J and the angle reference axis.
- **Incremental polar coordinates** always refer to the last programmed position of the tool.

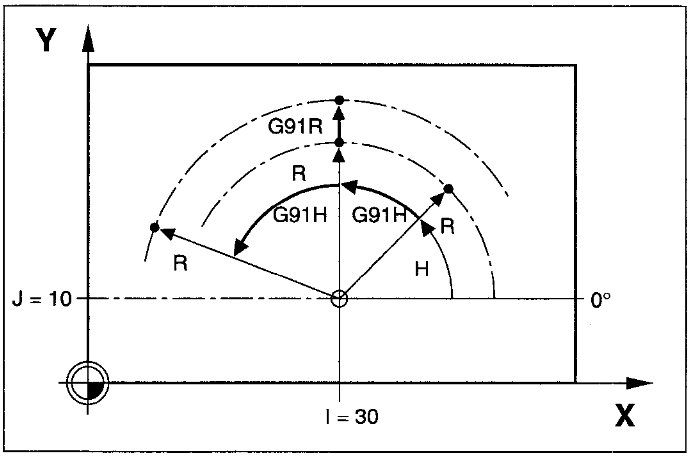
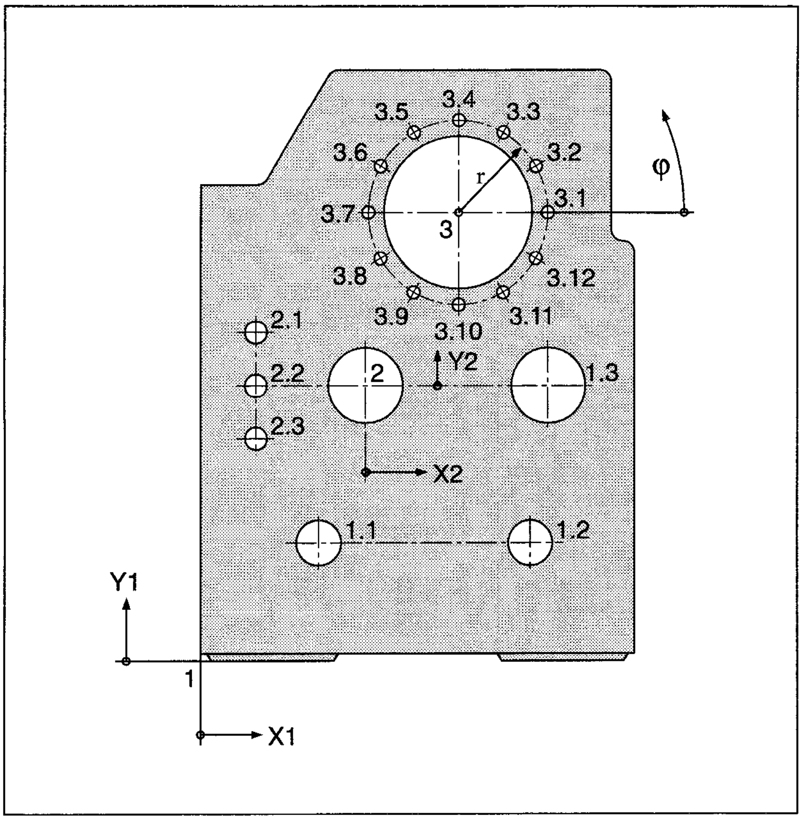


Fig. 1.19: Incremental dimensions in polar coordinates (designated with G91)

Example:

Workpiece drawing with coordinate dimensioning (according to ISO 129 or DIN 406, Part 11; Figure 179)



Coordinate origin		Dimensions in millimeters				
		Coordinates				
	Pos.	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 moving the tool or by moving the machine table on which the workpiece is fixed.

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

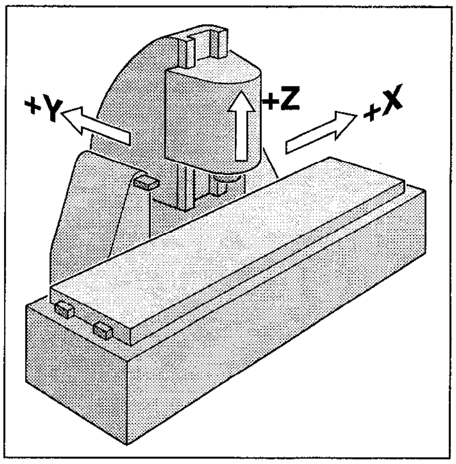


Fig. 1.21: On this machine the tool moves in the Y and Z axes, and the workpiece moves in the X axis

Position feedback encoders

Position feedback 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 the power is interrupted, the calculated position will no longer correspond to the actual position. But the TNC can re-establish this relationship when power is restored.

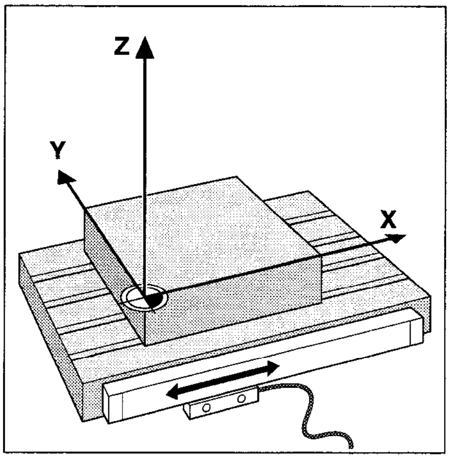


Fig. 1.22: Linear position encoder (here for the X axis)

Reference marks

The scales of the position encoders contain one or more reference marks. When a reference mark is crossed over, a signal is generated which identifies that position as the machine axis reference point. This allows the TNC to re-establish the relationship of displayed positions to machine axis positions.

If the position encoders have **distance-coded** reference marks, each axis need move no more than 20 mm (0.8 in.) for linear encoders, and 20° for angle encoders.

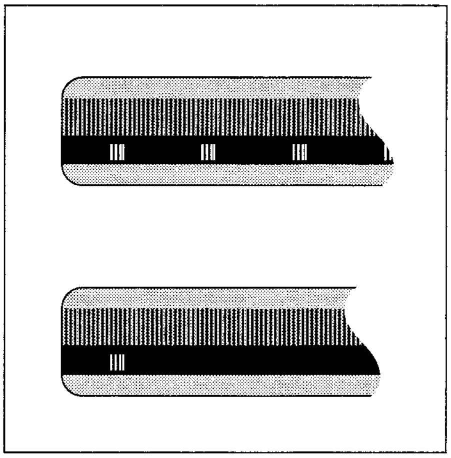


Fig. 1.23: Linear scales with distance-coded reference marks *(above)* and one reference mark *(below)*

1.3 Switch-On

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

Switch on the power supply for the control and the machine.
The following dialog then starts:

MEMORY TEST

The memory of the TNC is automatically checked.

POWER INTERRUPTED

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

TRANSLATE PLC PROGRAM

The PLC program of the TNC is automatically translated.

RELAY EXT. DC VOLTAGE MISSING

	Switch on the control voltage. The control checks the functioning of the EMERGENCY STOP circuit.

CROSS REFERENCE MARKS

X+ Y+	Cross over the reference marks manually using the machine direction keys, or
X ⋮	Cross the reference marks in several axes at once: Select appropriate axes (the selected axes are highlighted on the screen).
	Start the function for traversing the reference marks.

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

1.4 Graphics and Status Display

In the PROGRAMMING AND EDITING mode of operation the TNC shows a two-dimensional graphic of the programmed contour.

In the TEST RUN operating mode the TNC graphically simulates workpiece machining in:

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

The display mode is selected via soft keys.

The TNC graphic depicts the workpiece is if were being machined by a cylindrical end mill. The graphics cannot show rotary axis movements (error message).

With the fast internal image generation option, the TNC calculates the contour and displays a graphic only of the completed part.

Plan view

In this mode, contour height is indicated by image brightness. The higher the contour, the lighter the image.

Number of depth levels: 7

Plan view is the fastest of the three display modes.

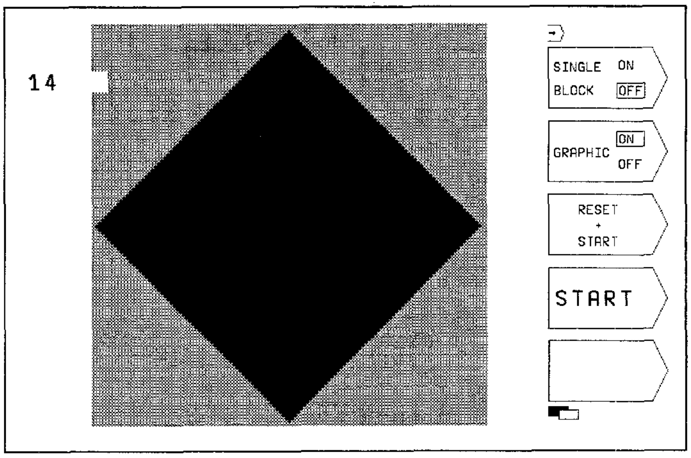


Fig. 1.24: Plan view

Projection in three planes

Here the program is displayed as in a technical drawing. The symbol at the left of the drawing indicates whether the display is in first angle or second angle projection according to ISO 6433, Part 1. The type of projection can be selected with MP 7310.

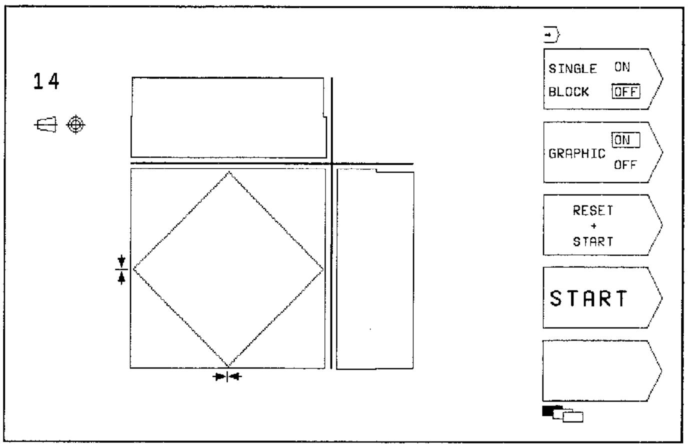


Fig. 1.25: Projection in three planes

Moving the sectional planes

The sectional planes can be moved to any position.

The position of the sectional plane is displayed on the screen while it is being moved.

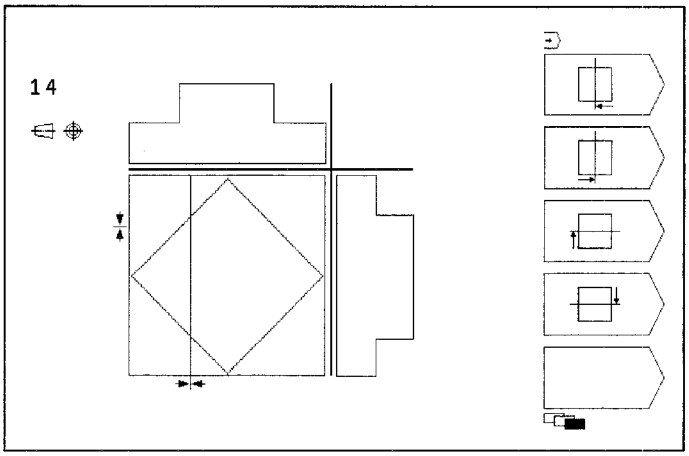


Fig. 1.26: Move sectional planes in the display mode *projection in three planes*

▽ or △	Shift the soft-key level.
or	Shift the vertical sectional plane to the right or left.
or	Shift the horizontal sectional plane upwards or downwards.

3D View

This mode displays the simulated workpiece as a solid model in three dimensional space.

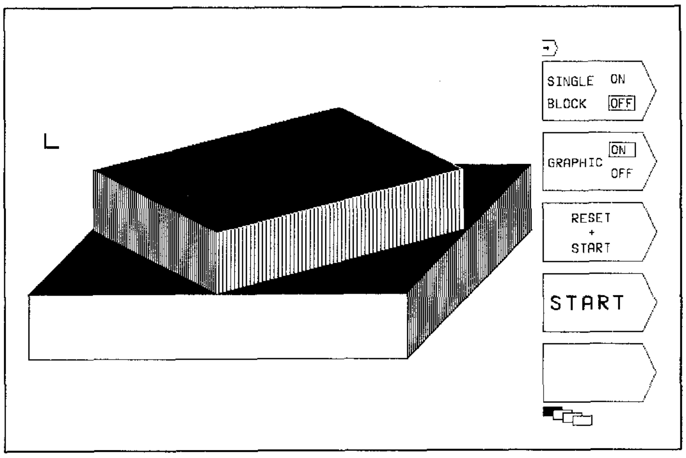


Fig. 1.27: TNC graphic display in 3D view

Rotating the 3D view

In the 3D view, the image can be rotated around the vertical axis.
The angle of orientation is indicated with a special symbol:

└	0° rotation
┘	90° rotation
┐	180° rotation
┌	270° rotation

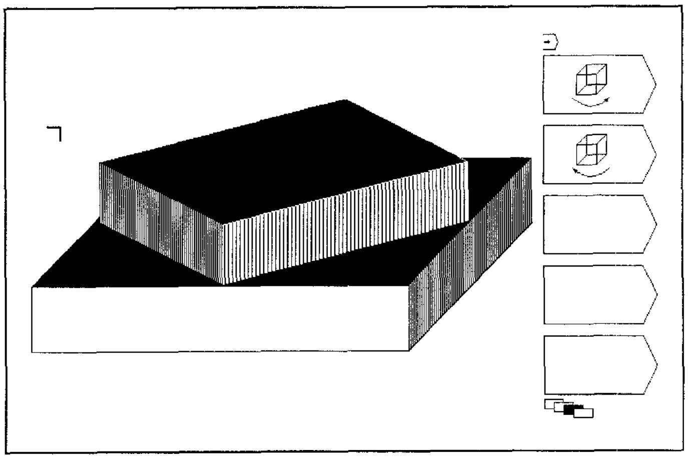


Fig. 1.28: Rotated 3D view

▽ or △	Shift the soft-key level.
or	Rotate the part by 90° in the horizontal plane.

Detail magnification

Details can only be magnified in the 3D view display mode.

The abbreviation MAGN appears on the screen to indicate that the image is magnified.

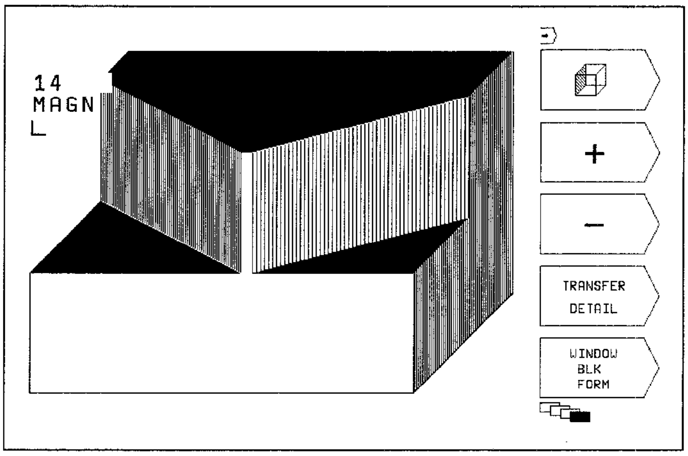
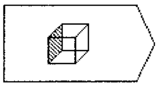


Fig. 1.29: Detail magnification of a 3D graphic

▽ or △	Shift the soft-key level.
	Select the side to be moved.
− or +	Reduce or enlarge workpiece.
TRANSFER DETAIL	Transfer detail.

Repeating graphic simulation

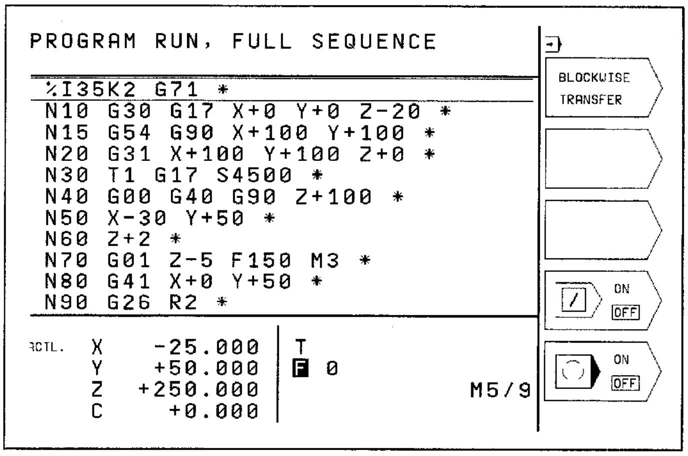
A part program can be graphically simulated as often as desired, either with the complete workpiece blank or with a detail of it.

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

Status display

The status display in the lower part of the screen provides the following information:

- Type of position values (ACTL, NOML, ...). The type of position values is selectable via MOD function)
- Axis is clamped (█ is shown in front of the axis)
- T: Number of the current tool
- X or Y or Z: Active tool axis
- S: Active spindle speed
- F: Active feed rate
- M: Active miscellaneous functions
- Spindle power indicated by a moving bar graphic
- TNC is in operation (indicated with ✻)
- On machines with different gear ranges: Gear range indicated behind the "/" character (depends on setting of machine parameter)
- The current mode of operation (above the soft keys)
- The number of available soft-key rows; active soft-key row is highlighted (beneath the soft keys)



```
PROGRAM RUN, FULL SEQUENCE

 %I35K2 G71 *
 N10 G30 G17 X+0 Y+0 Z-20 *
 N15 G54 G90 X+100 Y+100 *
 N20 G31 X+100 Y+100 Z+0 *
 N30 T1 G17 S4500 *
 N40 G00 G40 G90 Z+100 *
 N50 X-30 Y+50 *
 N60 Z+2 *
 N70 G01 Z-5 F150 M3 *
 N80 G41 X+0 Y+50 *
 N90 G26 R2 *

CTL.   X     -25.000 | T
       Y     +50.000 | F 0
       Z    +250.000 |               M5/9
       C      +0.000 |
```

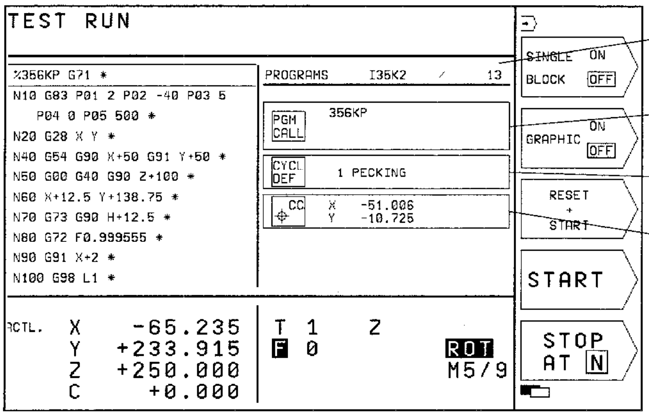
Fig. 1.30: Status display in a program run mode of operation

Additional status displays

The additional status displays contain further information on the program run. With the SELECT SCREEN LAYOUT key (see page 1-4) you can change the arrangement of the two status windows in the right half of the screen:

Additional status display	Soft key
General program information	TEXT PGM STATUS
Positions and coordinates	TEXT POS. STATUS
Tool information	TEXT TOOL STATUS
Coordinate transformations	TEXT COORD.TRANS. STATUS
Results of automatic tool measurement *(only available with conversational programming)*	TEXT TOOL PROBE STATUS

General program information



```
TEST RUN

%356KP G71 *                   PROGRAMS     I35K2    /     13
N10 G83 P01 2 P02 -40 P03 5
   P04 0 P05 500 *             PGM        356KP
N20 G28 X Y *                  CALL
N40 G54 G90 X+50 G91 Y+50 *
N50 G00 G40 G90 Z+100 *        CYCL
N60 X+12.5 Y+138.75 *          DEF      1 PECKING
N70 G73 G90 H+12.5 *
N80 G72 F0.999555 *             CC    X    -51.006
N90 G91 X+2 *                         Y    -10.725
N100 G98 L1 *

ACTL.  X     -65.235     T  1     Z
       Y    +233.915     F  0                 ROT
       Z    +250.000                          M5/9
       C      +0.000
```

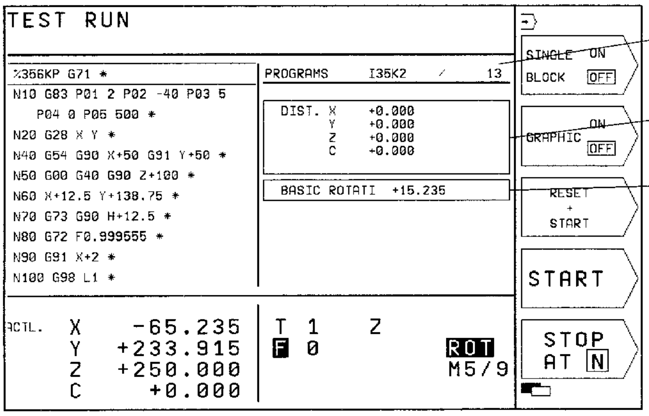
Active program / Current block number

Called programs

Active cycle

Circle center I,J (pole)

Positions and coordinates



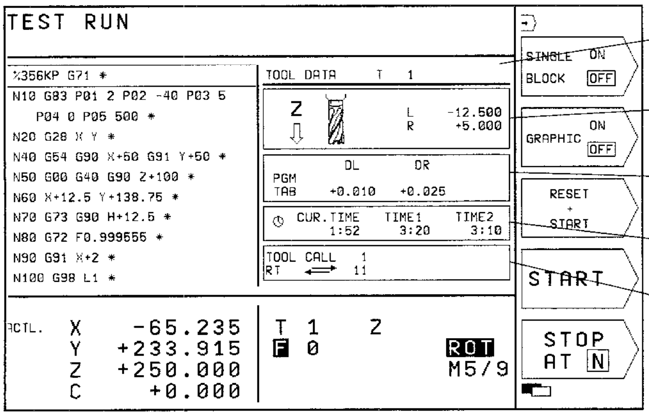
```
TEST RUN

%356KP G71 *                   PROGRAMS     I35K2    /     13
N10 G83 P01 2 P02 -40 P03 5
   P04 0 P05 500 *              DIST. X    +0.000
N20 G28 X Y *                         Y    +0.000
N40 G54 G90 X+50 G91 Y+50 *           Z    +0.000
N50 G00 G40 G90 Z+100 *               C    +0.000
N60 X+12.5 Y+138.75 *
N70 G73 G90 H+12.5 *            BASIC ROTATI   +15.235
N80 G72 F0.999555 *
N90 G91 X+2 *
N100 G98 L1 *

ACTL.  X     -65.235     T  1     Z
       Y    +233.915     F  0                 ROT
       Z    +250.000                          M5/9
       C      +0.000
```

Active program / Current block number

Additional position display

Display of active basic rotation

Tool information



```
TEST RUN

%356KP G71 *                   TOOL DATA     T   1
N10 G83 P01 2 P02 -40 P03 5
   P04 0 P05 500 *              Z                 L    -12.500
N20 G28 X Y *                                     R     +5.000
N40 G54 G90 X+50 G91 Y+50 *
N50 G00 G40 G90 Z+100 *                  DL       DR
N60 X+12.5 Y+138.75 *           PGM
N70 G73 G90 H+12.5 *            TAB    +0.010   +0.025
N80 G72 F0.999555 *
N90 G91 X+2 *                     CUR.TIME    TIME1    TIME2
N100 G98 L1 *                         1:52     3:20     3:10

                                TOOL CALL    1
                                RT           11

ACTL.  X     -65.235     T  1     Z
       Y    +233.915     F  0                 ROT
       Z    +250.000                          M5/9
       C      +0.000
```

Active tool

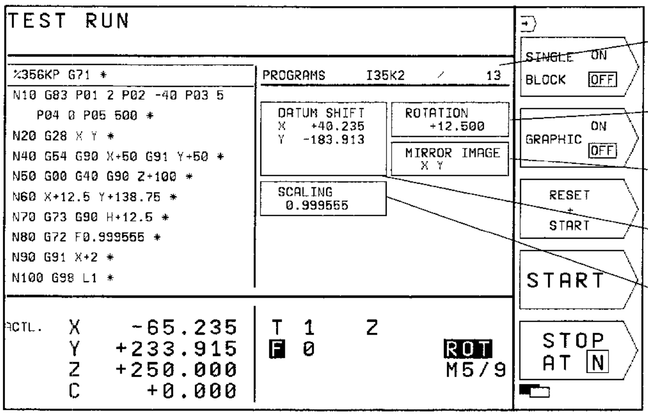
Active tool length and active tool radius

Oversizes from the tool table (TAB)

Tool life, maximum tool life and maximum tool life for tool call

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

Coordinate transformations



```
TEST RUN

%356KP G71 *
N10 G83 P01 2 P02 -40 P03 5
   P04 0 P05 500 *
N20 G28 X Y *
N40 G54 G90 X+50 G91 Y+50 *
N50 G00 G40 G90 Z+100 *
N60 X+12.5 Y+138.75 *
N70 G73 G90 H+12.5 *
N80 G72 F0.999555 *
N90 G91 X+2 *
N100 G98 L1 *

PROGRAMS     I35K2     /     13
DATUM SHIFT
X    +40.235
Y   -183.913
ROTATION
   +12.500
MIRROR IMAGE
  X Y
SCALING
 0.999555

ACTL.  X    -65.235   T 1    Z
       Y   +233.915   F 0          ROT
       Z   +250.000                M5/9
       C     +0.000
```

Active program / Current block number

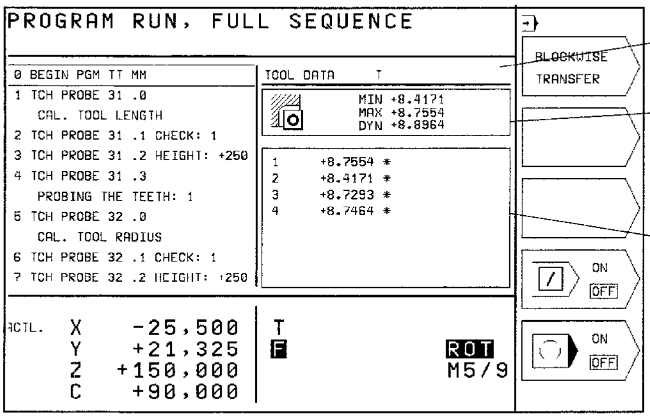
Active rotation angle defined in Cycle G73

Mirrored axes defined in Cycle G28

Active datum shift defined in Cycle G54

Active scaling factor defined in Cycle G72

Results of automatic tool measurement *(conversational programming only)*



```
PROGRAM RUN, FULL SEQUENCE

0 BEGIN PGM TT MM
1 TCH PROBE 31 .0
   CAL. TOOL LENGTH
2 TCH PROBE 31 .1 CHECK: 1
3 TCH PROBE 31 .2 HEIGHT: +250
4 TCH PROBE 31 .3
   PROBING THE TEETH: 1
5 TCH PROBE 32 .0
   CAL. TOOL RADIUS
6 TCH PROBE 32 .1 CHECK: 1
7 TCH PROBE 32 .2 HEIGHT: +250

TOOL DATA      T
          MIN +8.4171
          MAX +8.7554
          DYN +8.8964
1      +8.7554 *
2      +8.4171 *
3      +8.7293 *
4      +8.7464 *

ACTL.  X    -25,500   T
       Y    +21,325   F            ROT
       Z   +150,000                M5/9
       C    +90,000
```

Number of the tool measured

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

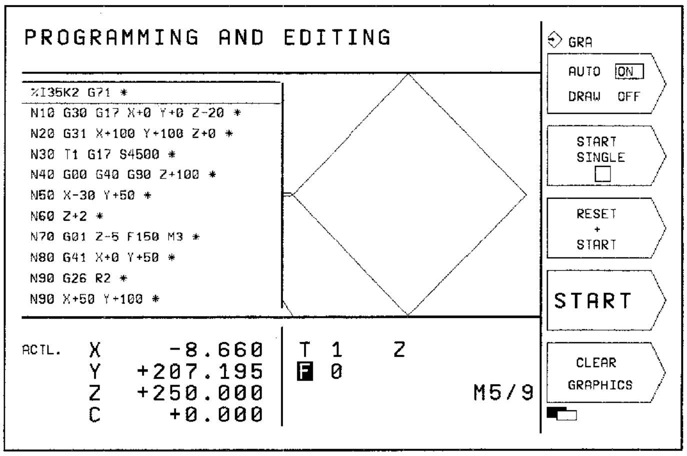
Cutting edge number with the corresponding measured value. If the measured value is followed by an asterisk (*), the allowable wear tolerance defined in the tool table was exceeded. If the measured value is followed by the letter **B**, the allowable break tolerance defined in the tool table was exceeded.

1.5 Interactive Programming Graphics

The TNC's interactive graphics generates the part contour as it is being programmed. Movements in the negative tool axis direction are displayed as a circle (circle diameter = tool diameter) to provide interactive graphics for drilling operations as well. In addition, the following functions are available:

- Detail magnification
- Detail reduction
- Clearing the graphic

The graphic functions are selected exclusively with soft keys.



PROGRAMMING AND EDITING

```
%I35K2 G71 *
N10 G30 G17 X+0 Y+0 Z-20 *
N20 G31 X+100 Y+100 Z+0 *
N30 T1 G17 S4500 *
N40 G00 G40 G90 Z+100 *
N50 X-30 Y+50 *
N60 Z+2 *
N70 G01 Z-5 F150 M3 *
N80 G41 X+0 Y+50 *
N90 G26 R2 *
N90 X+50 Y+100 *
```

```
ACTL.  X     -8.660 | T 1    Z
       Y   +207.195 | F 0
       Z   +250.000 |            M5/9
       C     +0.000 |
```

Fig. 1.31: Programming graphics

If you wish to work with the programming graphics, you must switch the screen layout to GRAPHICS or TEXT + GRAPHICS (see page 1-4).

Generating graphics during programming

▽ or △	If necessary, shift the soft-key level.
AUTO ON DRAW OFF	Select/deselect graphic generation during programming. The default setting is OFF.

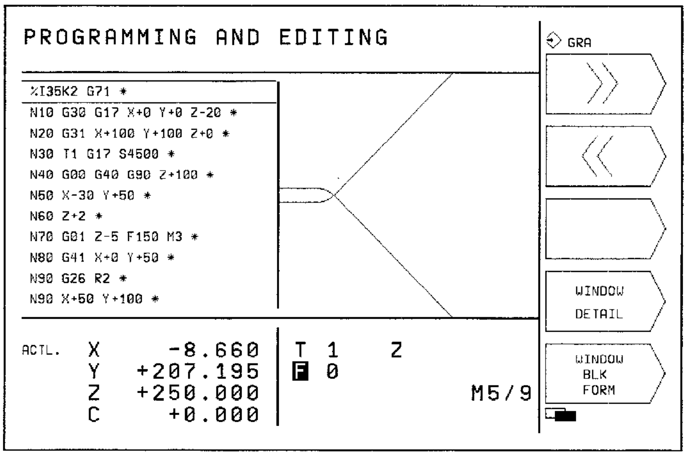
Generating graphics for an existing program

▽ or △	If necessary, shift the soft key level.
↑ or ↓	Select the desired block with the vertical cursor keys.
GOTO □ e.g. 4 7	Enter the desired block number, e.g. 47.
RESET + START	Generate a graphic from block 1 to the entered block.

Function	Soft key
Generate programming graphics blockwise	START SINGLE ☐
Generate a complete graphic or complete it after RESET + START	START
Interrupt programming graphics	STOP
Reset graphics	CLEAR GRAPHICS

The STOP soft key appears while the TNC generates the graphic.

Magnifying/reducing a detail



```
PROGRAMMING AND EDITING

%I35K2 G71 *
N10 G30 G17 X+0 Y+0 Z-20 *
N20 G31 X+100 Y+100 Z+0 *
N30 T1 G17 S4500 *
N40 G00 G40 G90 Z+100 *
N50 X-30 Y+50 *
N60 Z+2 *
N70 G01 Z-5 F150 M3 *
N80 G41 X+0 Y+50 *
N90 G26 R2 *
N90 X+50 Y+100 *

ACTL.  X      -8.660 | T 1    Z
       Y    +207.195 | F 0
       Z    +250.000 |              M5/9
       C      +0.000 |
```

Fig. 1.32: Detail from an interactive graphic

▽ or △	Shift the soft-key level.
>> or <<	Reduce or enlarge the frame overlay.
← → ↑ ↓	Shift the frame overlay.
ENT	Show the selected section.

1.6 Programs

The TNC can store up to 64 part programs at once. The programs can be written in HEIDENHAIN plain language dialog or ISO format.

Each program name can be up to 8 characters long (numbers or letters).

Program management

The program management function is called up with the PGM MGT soft key. With program management, programs can be input or output, protected, renamed and erased.

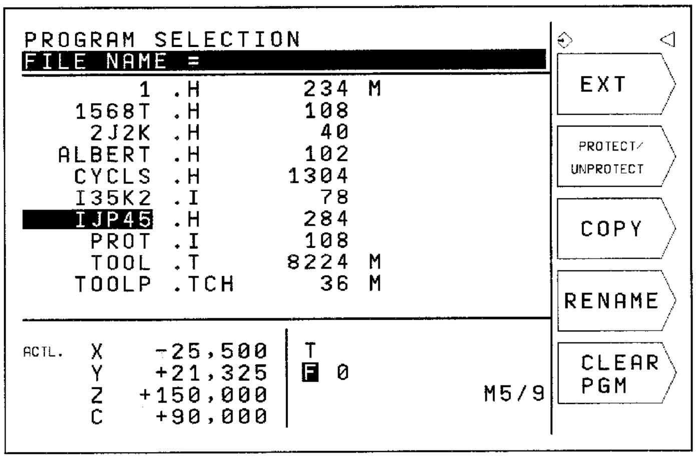
The individual functions are selected by pressing the corresponding soft keys.

Action	Operating mode	Call program management with ...
Create new program	◈	PGM MGT
Edit program	◈	PGM MGT
Input/output program	◈	PGM MGT EXT
Protect program	◈	PGM MGT PROTECT/ UNPROTECT
Rename program	◈	PGM MGT RENAME
Erase program	◈	PGM MGT CLEAR PGM
Test program	➲	PGM MGT
Execute program	➲ ⊟	PGM MGT

The program directory provides the following information:

- Program name
- Program type .H (HEIDENHAIN plain language)
  Program type .I (ISO)
  Program type .T (tool or pocket table)
- Program size in byte (1 byte = 1 character)
- Program status:

M:	Program is selected in operating mode EXECUTE
P:	Program is protected



```
PROGRAM SELECTION
FILE NAME =
       1 .H      234 M
   1568T .H      108
    2J2K .H       40
  ALBERT .H      102
   CYCLS .H     1304
   I35K2 .I       78
   IJP45 .H      284
    PROT .I      108
    TOOL .T     8224 M
   TOOLP .TCH     36 M

ACTL. X   -25,500  T
      Y   +21,325  F 0
      Z  +150,000            M5/9
      C   +90,000
```

Fig. 1.33: Program directory on the screen

Selecting, protecting, renaming and erasing programs

To select a program

PGM MGT	Call program management.
FILE NAME =	
↑ or ↓	Use cursor keys to select the program.
1 5	Enter the desired program name, for example 15.
ENT	Confirm your selection.

To protect a program

PGM MGT	Call program management.
FILE NAME =	
↑ or ↓	Use cursor keys to select the program.
PROTECT/ UNPROTECT	Protect the program. Protected status is indicated with a P.

To remove edit protection

PGM MGT	Call program management.
FILE NAME =	
↑ or ↓	Use cursor keys to select the program.
PROTECT/ UNPROTECT	Press the soft key to remove protection.
CODE NUMBER =	
8 6 3 5 7 ENT	Enter the code number 86357 and confirm with ENT. File protection is removed, the P disappears.

To rename a program

PGM MGT	Select program management.
FILE NAME =	
↑ or ↓	Use cursor keys to select the program.
RENAME	Rename the file.
DESTINATION FILE	
Type the new program name into the highlight in the screen headline and confirm with ENT.	

To erase a program

PGM MGT	Select program management.
FILE NAME =	
↑ or ↓	Use cursor keys to select the program.
CLEAR PGM	Press the CLEAR PROGRAM soft key.
YES	The file is erased.
NO	The file is not erased.

2 Manual Operation and Setup

2.1 Moving the Machine Axes

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

Traversing with the axis direction keys

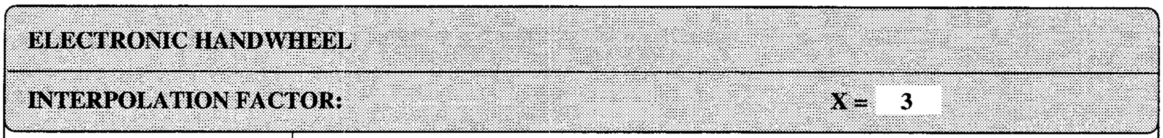
MANUAL OPERATION	
e.g. X+	Press the axis direction key and hold it down for as long as you want the axis to move.

You can move several axes at once in this way.

For continuous movement:

MANUAL OPERATION	
e.g. X+ I together	Press and hold the machine axis direction key, then press the NC start key. The axis continues to move after you release the keys.
0	To stop the axis, press the NC stop key.

Traversing with the electronic handwheel



ELECTRONIC HANDWHEEL

INTERPOLATION FACTOR: X = 3

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

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

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

Fig. 2.1: Interpolation factors and handwheel speed

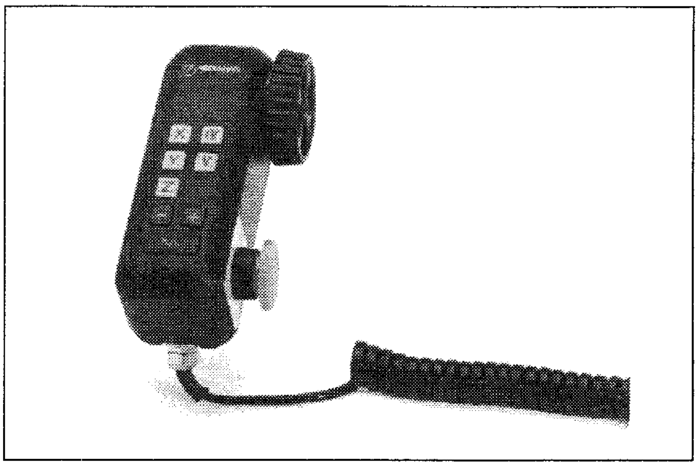


Fig. 2.2: HR 330 electronic handwheel

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

Working with the HR 330 electronic handwheel

The HR 330 is equipped with an enabling switch. The enabling switch is located on the side opposite from the handwheel and the emergency stop switch. You can move the machine axes only when the enabling switch is depressed.

- When the handwheel is magnetically attached to a flat surface the enabling switch is automatically depressed.
- Fasten the handwheel with its magnets to the machine in such a way that it cannot be operated unintentionally.
- When you remove the handwheel from the machine surface, make sure that you do not move the direction keys while the enabling switch is depressed.

Incremental jog positioning

With incremental jog positioning, a machine axis will move by a preset increment each time you press the corresponding machine axis direction key.

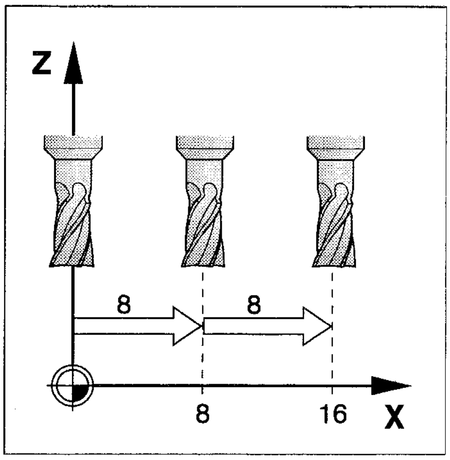


Fig. 2.3: Incremental jog positioning in the X axis

INCRE- ON MENT OFF	Select incremental jog positioning with the INCREMENT soft key.
JOG INCREMENT: 4 8	
e.g. 8 ENT	Enter the jog increment, for example 8 mm, or
e.g. 1	Set the jog increment with the soft keys, for example 1 mm.
e.g. Y+	Press the machine axis direction key as often as desired.

Press the INCREMENT soft key again to switch off incremental jog positioning.

Positioning with manual data input

Page 5-41 describes positioning by manually entering the target coordinates for the tool.

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

The following values can be entered and changed in the MANUAL OPERATION mode:

- Miscellaneous function M
- Spindle speed S
- Feed rate F (can be changed but not entered)

For part programs, these functions are entered directly in the PROGRAMMING AND EDITING operating mode.

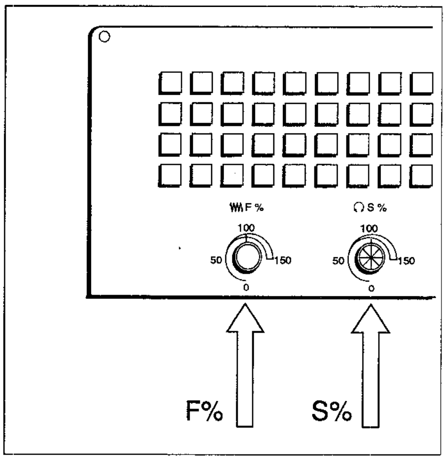


Fig. 2.4: Knobs for spindle speed and feed rate overrides

To enter the spindle speed S

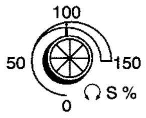
The machine manufacturer determines what spindle speeds are allowed on your TNC. Your machine manual provides more information on the available spindle speeds.

▽	Shift the soft-key row.
S	Select the S soft key to enter the spindle speed.
SPINDLE SPEED S =	
e.g. 1 0 0 0	Enter the spindle speed, for example 1000 rpm. Confirm the spindle speed S with the NC start key.

A miscellaneous function M starts spindle rotation at the entered speed S.

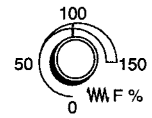
To change the spindle speed S

The spindle speed override will function only if your machine tool is equipped with a stepless spindle drive.

	Turn the spindle speed override knob: Adjust the spindle speed S to between 0 % and 150 % of the last entered value.

To change the feed rate F

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

	Turn the feed rate override knob: Adjust the feed rate from 0 % and 150 % of the last entered value.

To enter the miscellaneous function M

The machine manufacturer determines which miscellaneous functions are available on your TNC and what effects they have. Your machine manual provides more information on this subject.

▽	Shift the soft-key row.
M	Select the M soft key to enter the miscellaneous function.
MISCELLANEOUS FUNCTION	**M =**
e.g. 6	Enter the desired miscellaneous function M, for example M6.
	Activate the miscellaneous function M with the NC start key.

Chapter 11 provides an overview of the miscellaneous functions.

2.3 Setting the Datum Without a 3D Touch Probe

You fix a datum by setting the TNC 370 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 system from HEIDENHAIN (see page 2-15).

To prepare the TNC

Clamp and align the workpiece.

Insert the zero tool with known radius into the spindle.

	Select the MANUAL OPERATION mode.

Ensure that the TNC is showing actual position values (see page 10-6).

Setting the datum in the tool axis

Protective measure:
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 the desired datum by the value *d*.

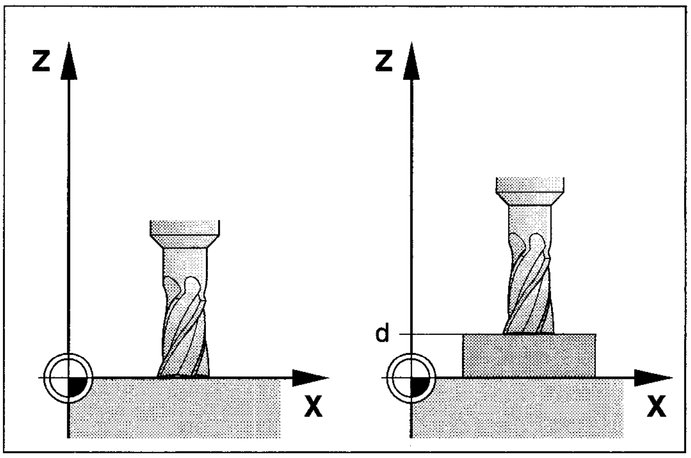


Fig. 2.5: Datum setting in the tool axis; right with protective shim

Move the tool until it touches the workpiece surface.

Z	Select the tool axis.

DATUM SET Z =

e.g. 0 ENT	For a zero tool: Set the display to Z = 0 or enter thickness d of the shim.
e.g. 5 0 ENT	For a preset tool: Set the display to the length L of the tool, for example Z = 50 mm, or enter the sum Z = L + d.

Setting the datum in the working plane

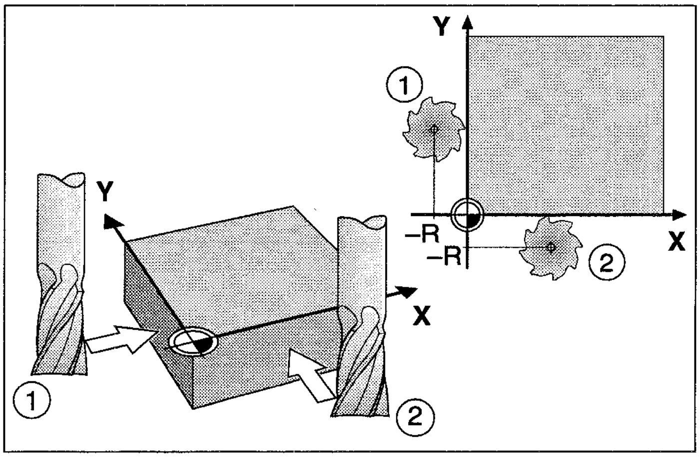


Fig. 2.6: Setting the datum in the working plane; plan view (upper right)

Move the zero tool until it touches the side of the workpiece.

e.g. X	Select the axis.

DATUM SET X=

e.g. – 5 ENT	Enter the position of the tool center (here X = –5 mm) in the selected axis. Be careful to enter the correct algebraic sign.

Repeat the process for all axes in the working plane.

2.4 3D Touch Probes

3D Touch probe applications

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

- Compensating workpiece misalignment (basic rotation)
- Datum setting
- Measuring:
  - Lengths and workpiece positions
  - Angles
  - Radii
  - Circle centers
- Measurements under program control

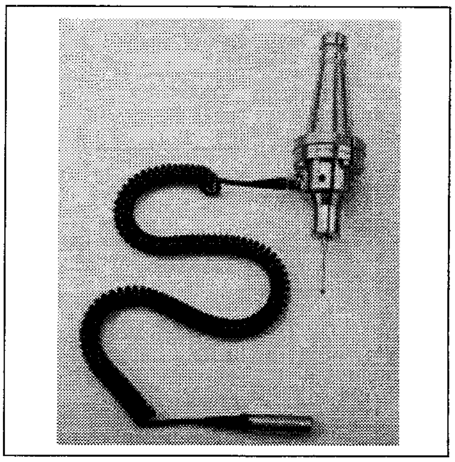


Fig. 2.7: HEIDENHAIN TS 220 3D-touch probe

The TNC must be specially prepared by the machine tool builder for the use of a 3D touch probe.

After you press the NC start key, the touch probe begins executing the selected probe function. The machine manufacturer sets the feed rate F at which the probe approaches the workpiece (MP 6120). When the 3D touch probe contacts the workpiece, it

- transmits a signal to the TNC:
  The coordinates of the probed position are stored
- stops moving
- returns to its starting position in rapid traverse.

If the stylus in not deflected within the distance defined in MP6130, the TNC displays an error message.

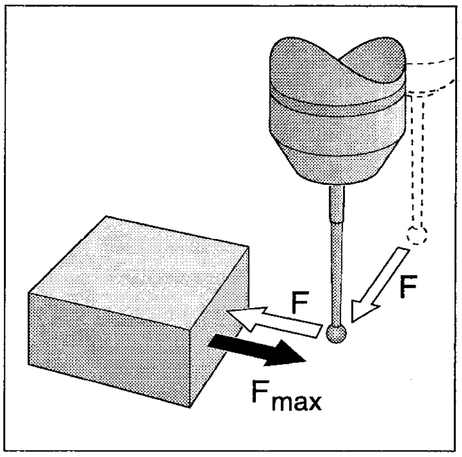


Fig. 2.8: Feed rates during probing

To select the touch probe functions

▶ MANUAL OPERATION

or

▶ ELECTRONIC HANDWHEEL

TOUCH PROBE	Select the touch probe functions.

Function	Soft key
Compensate workpiece misalignment	PROBING ROT
Set datum in any axis	PROBING POS
Set a corner as datum	PROBING P
Set the datum at a circle center	PROBING CC
Calibrate the effective length	CAL L
Calibrate the effective radius	CAL R

Calibrating the 3D touch probe

The touch probe system must be calibrated

- for commissioning
- after a stylus breaks
- when the stylus is changed
- when the probe feed rate is changed
- in case of irregularities, such as those resulting from machine heating.

During calibration, the control finds the "effective" length of the stylus and the "effective" radius of the ball tip. To calibrate the 3D touch probe, clamp a ring gauge with known height and known internal radius to the machine table.

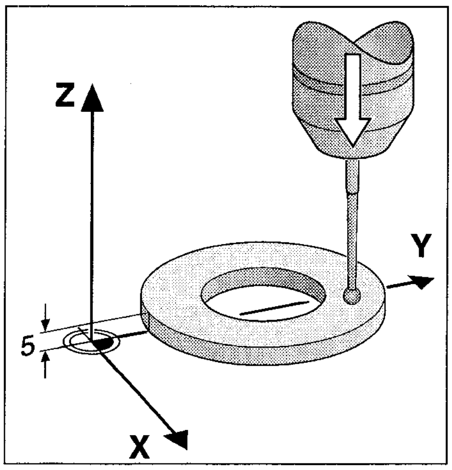


Fig. 2.9: Calibrating the touch probe length

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.

CALIBRATION EFFECTIVE LENGTH

Z+ Z–

TOOL AXIS = Z

e.g. Z	If necessary, enter the tool axis, for example Z.
↓	Move the highlight to DATUM.
e.g. 5	Enter the height of the ring gauge, for example 5 mm.

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

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

	The 3D 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.

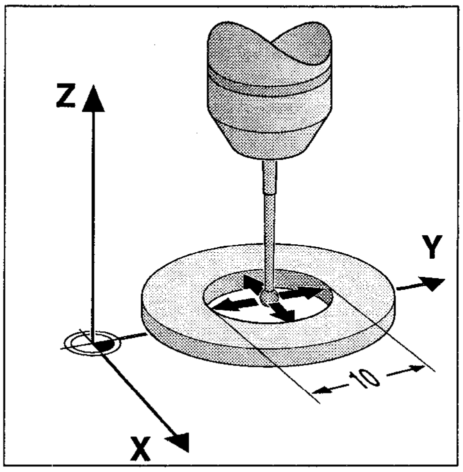


Fig. 2.10: Calibrating the touch probe radius

CAL R	Select the calibration function for the ball-tip radius.

CALIBRATION EFFECTIVE RADIUS

X+ X– Y+ Y–

↓	Select RADIUS RING GAUGE.

RADIUS RING GAUGE = 0

5 ENT	Enter the radius of the ring gauge, for example 5 mm.
4 x	The 3D touch probe contacts one position on the bore hole for each axis direction.

Displaying calibration values

The TNC 370 stores the effective length and radius of the 3D touch probe for use whenever the touch probe is needed again. The stored values are displayed the next time the calibration function is called.

Compensating workpiece misalignment

The TNC 370 electronically compensates workpiece misalignment by computing what is called a *basic rotation.*

Set the ROTATION ANGLE to the angle at which a workpiece surface should be oriented with respect to the angle reference axis (see page 1-17).

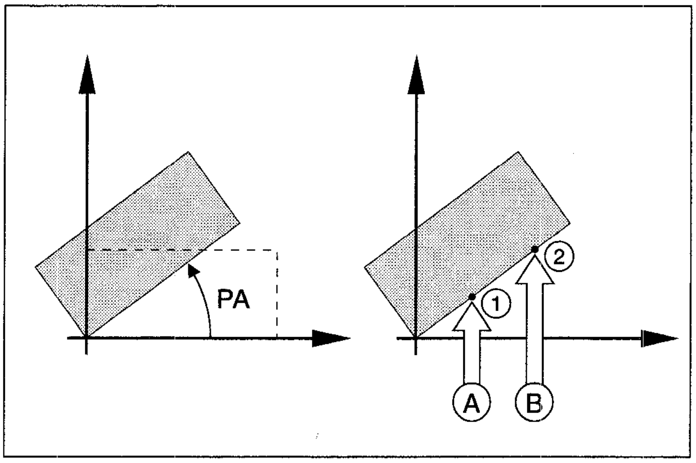


Fig. 2.11: Basic rotation of a workpiece *(left)*, probing procedure for compensation *(right)*. The dashed line is the nominal position; the angle *PA* is being compensated.

PROBING ROT	Select the probe function for compensating workpiece misalignment.

BASIC ROTATION

X+ X– Y+ Y–

ROTATION ANGLE =

e.g. 0 ENT	Enter the nominal value of the ROTATION ANGLE.

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

X+ X– Y+ Y–

← or →	Select the probe direction.
	Probe the workpiece.

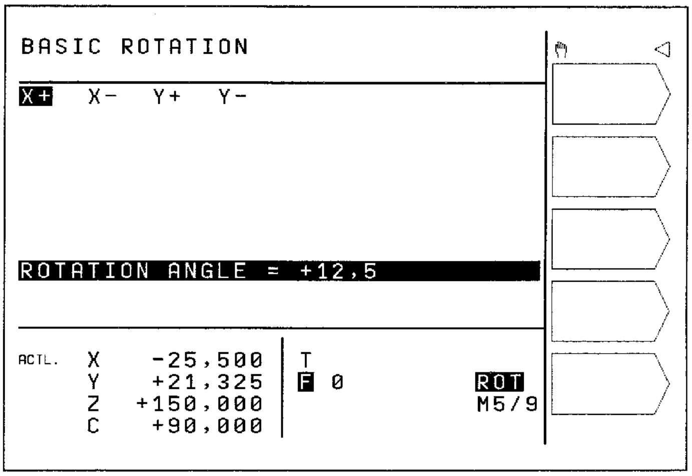
Move the ball tip to a starting position Ⓑ near the second touch point ②.

	Probe the workpiece.

A basic rotation is kept in nonvolatile storage and is effective for all subsequent program runs and graphic test runs.

Displaying basic rotation

The angle of the basic rotation is shown in the rotation angle display. When a basic rotation is active the abbreviation ROT is highlighted in the status display.



```
BASIC ROTATION
X+   X-   Y+   Y-

ROTATION ANGLE  =  +12,5

ACTL.  X     -25,500 | T
       Y     +21,325 | F 0          ROT
       Z    +150,000 |              M5/9
       C     +90,000 |
```

Fig. 2.12: Displaying the angle of an active basic rotation

To cancel a basic rotation

Select the probe function for compensating workpiece misalignment again.

ROTATION ANGLE =

0 ENT	Set the ROTATION ANGLE to 0.
END	Terminate the probing function.

2.5 Setting the Datum with the 3D Touch Probe

The following functions for setting the datum on an aligned workpiece are listed in the TCH PROBE menu and are selected with the appropriate soft keys.

To set the datum in a specific axis

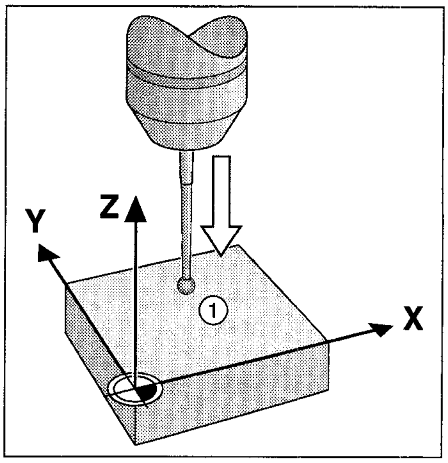


Fig. 2.13: Probing for the datum in the Z axis

Select the probe function for datum setting in any axis.

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

```
WORKPIECE SURFACE = DATUM

X+    X–    Y+    Y–    Z +   Z–
```

← or →	Select the probing direction and the axis in which you wish to set the datum, for example Z in the Z– direction.
	Probe the workpiece.
e.g. 0 ENT	Enter the nominal coordinate of the DATUM and confirm your entry.

Setting a corner as datum

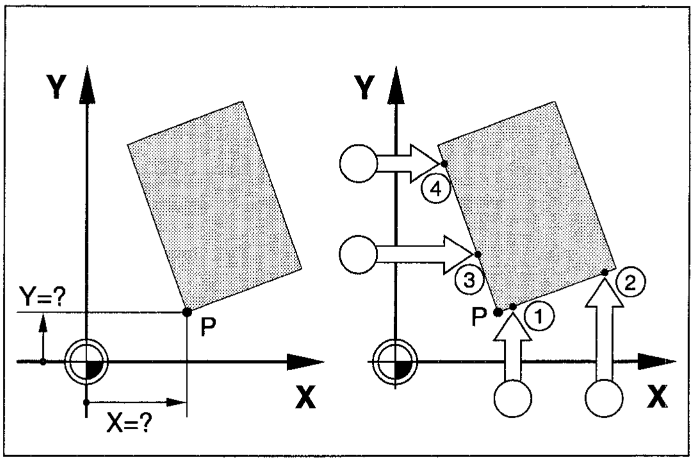


Fig. 2.14: Probing procedure for finding the coordinates of the corner P

Select the probe function for setting a corner as datum.

To use the points that were just probed for a basic rotation

TOUCH POINTS OF BASIC ROTATION?

YES	Transfer the touch point coordinates to memory.

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

CORNER = DATUM

X + X – Y + Y –

← or →	Select the probing direction.
	Probe the workpiece.

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

	Probe the workpiece.

DATUM X =

e.g. 0 ENT	Enter the first coordinate of the datum point, for example in the X axis. Confirm your entry.

.
.
.

⋮

↓	Select the second coordinate.

DATUM Y =

e.g. 0 ENT	Enter the second coordinate of the datum, for example in the Y axis. Confirm your entry.

END □	Terminate the probe function.

If you do not wish to use points that were just probed for a basic rotation

TOUCH POINTS OF BASIC ROTATION?

NO	Ignore the dialog prompt.

Probe both workpiece sides twice.

Enter the datum coordinates.

Setting the datum at a circle center

With this function you can set the datum at the center of bore holes, circular pockets, cylinders, journals, circular islands, etc.

Inside circle

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

For incomplete circles (circular arcs) you can choose the appropriate probing direction.

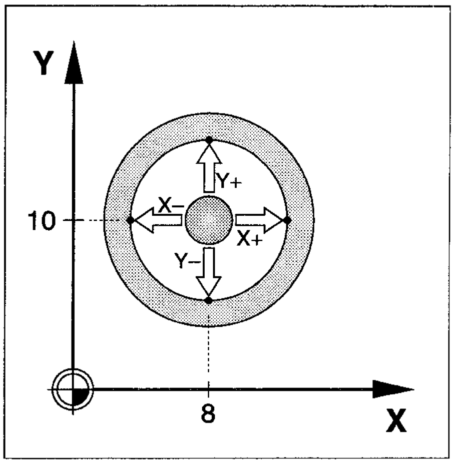


Fig. 2.15: Probing an inside cylindrical surface to find the center

Select the probe function for datum setting at a circle center

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

CIRCLE CENTER = DATUM

X + X – Y + Y –

4 x	The probe touches four points on the inside of the circle.

DATUM X =

e.g. 8 ENT	Enter the first coordinate of the circle center, for example in the X axis. Confirm your entry.
↓	Select the second coordinate.

DATUM Y =

e.g. 1 0 ENT	Enter the second coordinate of the circle center, for example in the Y axis. Confirm your entry.
END □	Terminate the probe function.

Outside circle

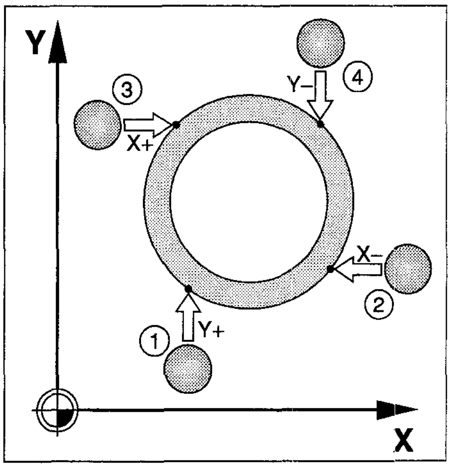


Fig. 2.16: Probing an outside cylindrical surface to find the center

Select the probe function for datum setting at a circle center.

Move the touch probe to a starting position ① near the first touch point outside of the circle.

CIRCLE CENTER = DATUM

X+ X– Y+ Y–

← or →	Select the probing direction.
	Probe the workpiece.

Repeat the probing process for points ②, ③ and ④ (see Fig. 2.16).

Enter the coordinates of the circle center.

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

2.6 Measuring with the 3D Touch Probe

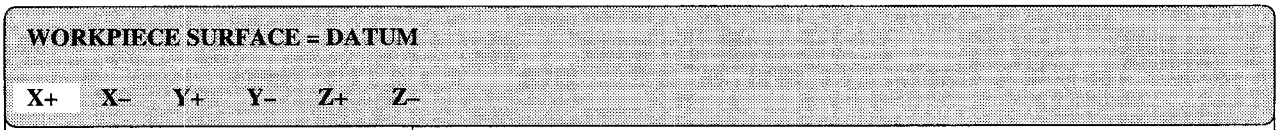
With the 3D touch probe system you can determine

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

Finding the coordinate of a position on an aligned workpiece

Select the probe function for datum setting in any axis.

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



```
WORKPIECE SURFACE = DATUM

X+   X–   Y+   Y–   Z+   Z–
```

← or →	Select the probing direction and the axis in which you wish to find the coordinate.
□	Probe the workpiece.

The TNC displays 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 "Setting a corner as datum."

The TNC displays the coordinates of the probed corner as DATUM.

Measuring workpiece dimensions

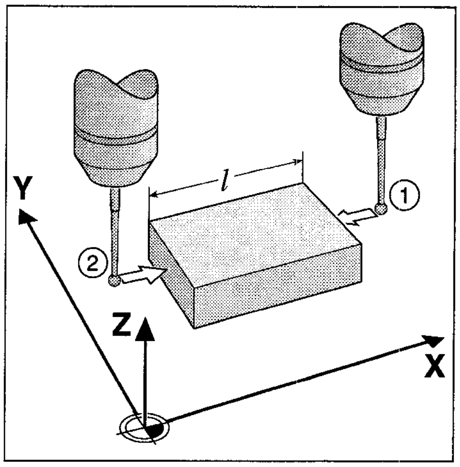


Fig. 2.17: Measuring lengths with the 3D touch probe

Select the probe function for setting the datum in any axis.

Move the touch probe to a starting position near the first touch point ①.

WORKPIECE SURFACE = DATUM

X+ X– Y+ Y– Z+ Z–

← or →	Select the probing direction with the arrow keys.
⎆	Probe the workpiece.

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

DATUM X=

0 ENT	Set the DATUM to 0 and confirm your entry.
END □	Terminate the dialog.

Re-select the probe function for datum setting in any axis.

Move the touch probe to a starting position ② near the second touch point.

.
.
.

⋮

WORKPIECE SURFACE = DATUM

X+ X– Y+ Y– Z+ Z–

← or →	Select the probing direction with the arrow keys - same axis as for ①.
	Probe the workpiece.

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

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

Select the probe function for datum setting in any axis.

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 3D touch probe system 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:

Select the probe function for compensating workpiece misalignment.

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 "Compensating workpiece misalignment").

⋮

.
.
.

The angle between the angle reference axis and the side of the workpiece appears as the ROTATION ANGLE in the BASIC ROTATION function.

Cancel the basic rotation.

Restore the previous basic rotation by setting the ROTATION ANGLE to the value that you wrote down previously.

To measure the angle between two sides of a workpiece:

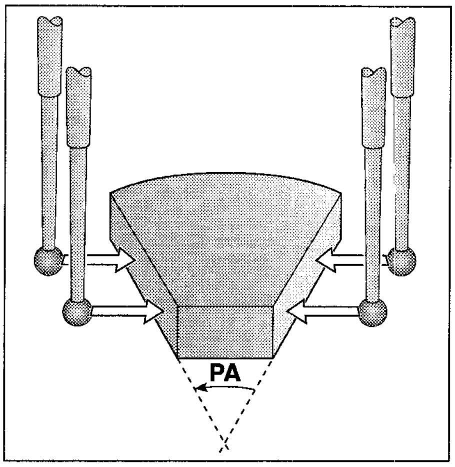


Fig. 2.18: Measuring the angle between two sides of a workpiece

Select the probe function for compensating workpiece misalignment.

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!

The angle PA between the workpiece sides appears as the ROTATION ANGLE in the BASIC ROTATION function.

Cancel the basic rotation.

Restore the previous basic rotation by setting the ROTATION ANGLE to the value that you wrote down previously.

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 incompatibility
- Missing data
- Impossible jumps

To do a test run

START	Test the entire program.
STOP AT N	Test the program up to a certain block number.

Test run features

Function	Soft key
Test run in Full Sequence or Single Block	SINGLE BLOCK ON OFF
Test run with or without graphics	GRAPHIC ON OFF
Start test run from beginning of program	RESET + START
Start test run from current block	START
Test run up to block N	STOP AT N

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 execute each block separately by pressing the NC start key.

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 (DNC operation)
- Checking or changing Q parameters

To run a part program

- Clamp the workpiece to the machine table.
- Set the datum.
- Select the program.

PROGRAM RUN / FULL SEQUENCE

or

PROGRAM RUN / SINGLE BLOCK

Select the part program.

GOTO 0 ENT	Select the first block of the program.
	Run the part program.
Only in mode PROGRAM RUN / SINGLE BLOCK repeatedly	Run each block of the part program separately.

The feed rate and spindle speed can be changed with the override knobs.

Interrupting machining

There are several ways to interrupt a program run:

- Programmed interruptions
- External STOP key
- Switching to PROGRAM RUN / SINGLE BLOCK
- EMERGENCY STOP button

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

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 functions M00, M01, M02 or M30
- Miscellaneous function M06, if the machine tool builder has assigned a stop function

To interrupt or abort machining:

The block which the TNC 370 is currently executing is not completed:

0	Interrupt machining.

The ✻ symbol in the status display blinks.

The part program can be aborted with the STOP soft key:

STOP	Abort program run.

The ✻ symbol disappears from the status display.

To interrupt machining by switching to the PROGRAM RUN / SINGLE BLOCK operating mode:

The program run is interrupted at the end of the current block.

▣	Select PROGRAM RUN / SINGLE BLOCK.

Resuming program run after an interruption

When a program run is interrupted the control stores the following information:

- 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 repeat
- The number of the last block that calls a subprogram or a program section repeat

- If you have interrupted machining with the STOP soft key, you can jump to another block with GOTO and resume machining from there.
- If you select block 0, the TNC will reset all stored data (such as tool data, etc.).
- If a program run was interrupted during a program section repeat, GOTO can only be used to jump to other blocks within the program section repeat.

Resuming program run with the START button

You can resume program run by pressing the NC start key if the program was interrupted in one of the following ways:

- Pressing the NC stop key
- A programmed interruption
- Pressing the EMERGENCY STOP button (machine-dependent function)

Resuming program run after an error

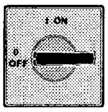
- If the error message is not blinking:

Remove the cause of the error.

CE Clear the error message from the screen.

Restart the program.

- If the error message is blinking:



Switch off the TNC 370 and the machine.

Remove the cause of the error.

Restart the program.

- If you cannot correct the error:

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

3.3 Blockwise Transfer: Executing Long Programs

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.

To prepare for blockwise transfer:

- Prepare the data interface.
- Configure the data interface with the MOD function (see page 10-5).
- If you wish to transfer a part program from a PC, adapt the TNC and PC to each other (see pages 9-4 and 11-3).
- Ensure that the transferred program meets the following requirements:
  - The highest block number must not exceed 65534. However, the block numbers can repeat themselves as often as necessary.
  - All programs called from the transferred program must be present in the control's memory.
  - The transferred program must not contain:
    Subprograms
    Program section repeats
    The function D15:PRINT
  - The TNC can store up to 20 G99 blocks.

PROGRAM RUN / SINGLE BLOCK

or

PROGRAM RUN / FULL SEQUENCE

BLOCKWISE TRANSFER	Select the function for blockwise transfer.

PROGRAM NUMBER

e.g. 1 0 ENT	Enter the program number and start data transfer.
	Execute the transferred program blocks.

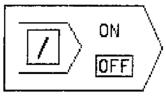
If data transfer is interrupted, press the NC start key again.

3.4 Skipping Blocks During Program Run

Blocks that were programmed with a "/" sign can be ignored during a program run.

PROGRAM RUN / FULL SEQUENCE

Select the part program.



Set the soft key to OFF to execute programs with the "/" blocks, or set the soft key to ON to execute programs without the "/" blocks.

This function is not effective with G99 blocks.

3.5 Optional Interruption of a Program Run

The TNC interrupts the program run at blocks programmed with M01 if the optional stop soft key is set to ON.

PROGRAM RUN / FULL SEQUENCE

Select the part program.

Set the soft key to OFF to disable program run interruption at M01, or set the soft key to ON for program run interruption at M01.

4 Programming

4 Programming

In the PROGRAMMING AND EDITING mode of operation you can

- create files
- edit files
- erase files.

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

4.1 Editing Part Programs

Layout of a program

A part program consists of individual program blocks.

The TNC numbers the blocks in ascending order. The block number increment is set in MP 7220 (see page 11-5). Program blocks contain units of information called "words."

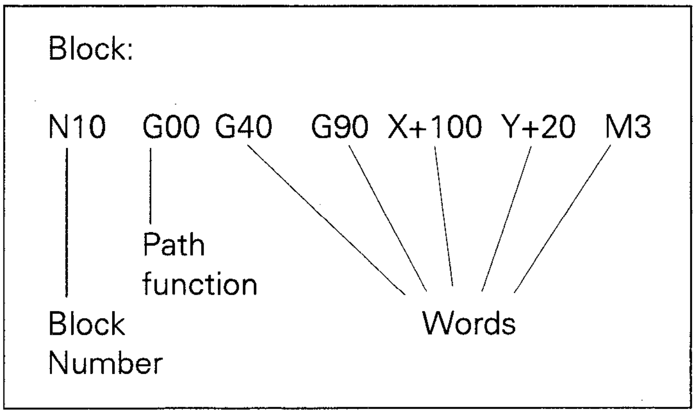


Fig. 4.1: Program blocks contain words of specific information

Function	Key
Continue the dialog	ENT
Skip dialog question	NO ENT
End the dialog immediately	END □
Abort the dialog (changes are canceled)	DEL

Editing functions

Editing means entering or changing commands and information for the TNC.

The TNC enables you to

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

Types of input

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	Block number 10 is shown between two horizontal lines.

- To move one block forward or backward:

↑ or ↓	Press the vertical arrow keys.

- To select individual words in a block:

← or →	Press the horizontal arrow keys.

- To find the same word in other blocks:

← or →	Select the word in the block.
↑ or ↓	Jump to the same word in other blocks.

Inserting blocks

Additional program blocks can be inserted behind any existing block (but not behind the N9999 block).

↑ or ↓ / GOTO □	Select the block in front of the desired insertion.
N e.g. 3 5 ENT	Program the 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 arrow key to remove the highlight from the block or confirm the change with the END key. You can also add words by moving the cursor with the horizontal arrow keys to the block you wish to change.

Erasing blocks and words

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

4.2 Tools

Each tool is identified by a number.

The tool data, consisting of the:

- Length L, and
- 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 program TOOL.T.

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

Determining tool data

Tool number

Each tool is designated with a number between 0 and 254.

Tool number 0 is defined as having length L = 0 and radius R = 0. In tool tables, T0 should also 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

Applications:

- Undersize in the tool table for wear
- Oversize in the tool table, for example as a finishing allowance during roughing.

Delta values can be numerical values or the value 0. The maximum permissible oversize or undersize is +/- 99.999 mm.

Determining tool length with a zero tool

For the sign of the tool length L:

$L > L_0$	A positive value means the tool is longer than the zero tool.
$L < L_0$	A negative value means the tool is shorter than the zero tool.

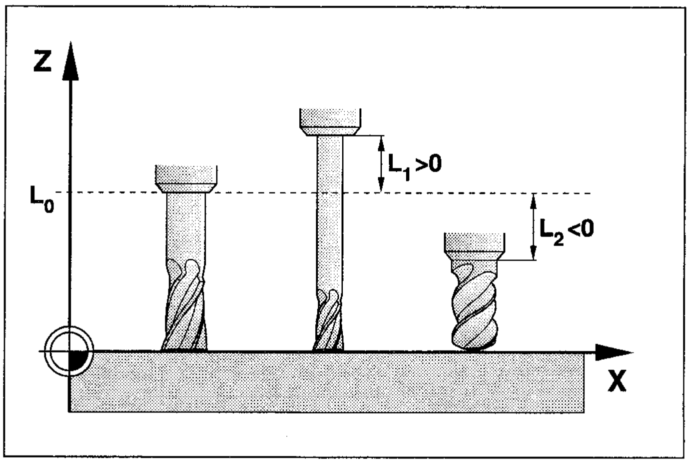


Fig. 4.2: Tool lengths can be given as the difference from the zero tool

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.

Write the value down 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 for each tool in the part program:

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

To enter tool data in the program block:

G 9 9 ENT ▶

▶ TOOL NUMBER	
e.g. 5 ENT	Designate the tool with a number, for example 5.

TOOL LENGTH L	
1 0 ENT	Enter the compensation value for the tool length, for example L = 10 mm.

TOOL RADIUS R	
e.g. 5 END ☐	Enter the tool radius, for example R = 5 mm.

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

- You can save the tool length and tool radius as an actual position by pressing the ACTUAL POS. X, ACTUAL POS. Y or ACTUAL POS. Z soft key.
- If you are using program TOOL.T, enter tool numbers that are larger than 99 in the G99 block.

Entering tool data in program TOOL.T

The data for all tools can be entered in a common tool table. The number of tools in the table is selected through the machine parameter MP 7260. With MP7266 you can determine which data can be entered in the tool table as well as the sequence in which they appear (see page 11-5).

If your machine uses an automatic tool changer or if you are working with the automatic tool measurement function, the tool data must be stored in the tool table.

Editing the tool table (program TOOL.T)

⇨ ▶ **PROGRAMMING AND EDITING**

PGM MGT	Call the program directory.

▼

PROGRAM NAME =

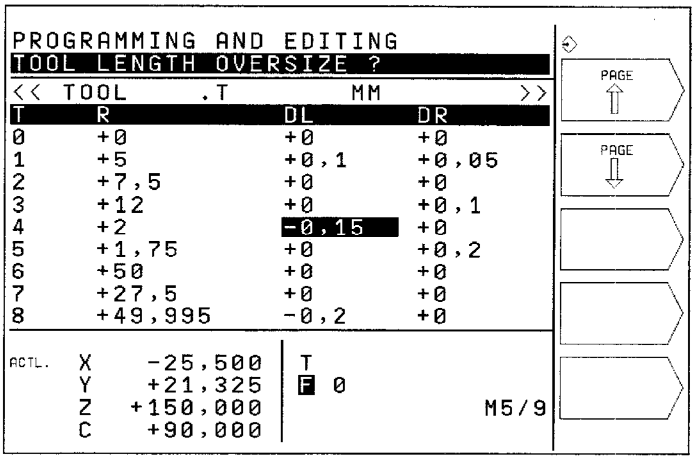
↓ ENT	Select the tool table.

Dialog guidance in the tool table	Key
Confirm the entered value, select the next column in the table	ENT
Clear an incorrect numerical entry	CE
Recall the last stored value	DEL
Move the highlight one column to the left/right	← / →
Move the highlight one line up/down	↑ / ↓
Page one screen upward	PAGE ⇧
Page one screen downward	PAGE ⇩
Answer yes to a dialog question	YES
Answer no to a dialog question	NO

Entering tool data in tables

You can enter the following information in the tool table:

- Tool lengths and radii : L, R
- Oversizes (delta values) for tool radius and length: DR, DL
- Tool name: NAME (maximal 16 characters)
- Maximum tool life and current tool age: TIME1, TIME2, CUR.TIME
- Number of a Replacement Tool: RT
- Tool Lock: TL
- Tool comment: DOC
- Information on this tool for the Programmable Logic Control (PLC), which integrates the control to the machine: PLC

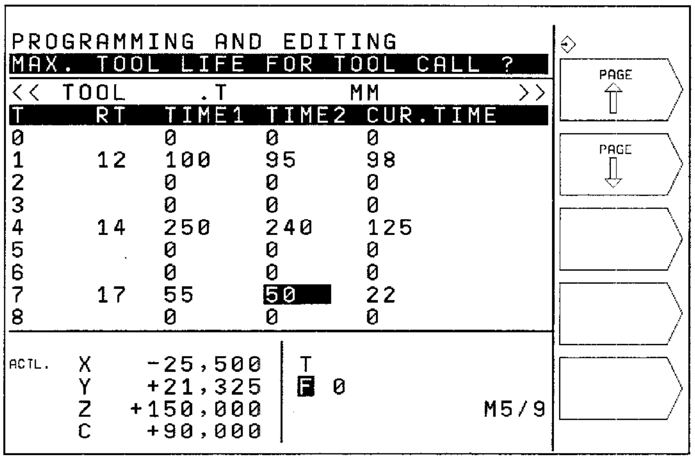


```
PROGRAMMING AND EDITING
TOOL LENGTH OVERSIZE ?
<< TOOL       .T          MM          >>
T        R                DL          DR
0        +0               +0          +0
1        +5               +0,1        +0,05
2        +7,5             +0          +0
3        +12              +0          +0,1
4        +2               -0,15       +0
5        +1,75            +0          +0,2
6        +50              +0          +0
7        +27,5            +0          +0
8        +49,995          -0,2        +0

ACTL.  X    -25,500  |  T
       Y    +21,325  |  F  0
       Z   +150,000  |                M5/9
       C    +90,000  |
```

Fig. 4.3: Section of a tool table

The TNC needs the following tool data for automatic tool measurement (only available with conversational programming):

- Number of cutting edges for tool measurement: CUT.
- Tolerance for wear in the tool length for automatic tool measurement: LTOL
- Tolerance for wear in the tool radius for automatic tool measurement: RTOL
- Cutting direction of the tool for dynamic tool measurement: DIRECT.
- Offset of the tool center from stylus center for automatic tool length measurement: TT:R-OFFS Default setting: tool radius R
- Offset of the tool end from stylus top for automatic tool radius measurement: TT:L-OFFS Default setting: 0
- Breakage tolerance in the tool length for automatic tool measurement: LBREAK
- Breakage tolerance in the tool radius for automatic tool measurement: RBREAK



```
PROGRAMMING AND EDITING
MAX. TOOL LIFE FOR TOOL CALL ?
<< TOOL       .T          MM          >>
T        RT    TIME1  TIME2  CUR.TIME
0              0      0      0
1        12    100    95     98
2              0      0      0
3              0      0      0
4        14    250    240    125
5              0      0      0
6              0      0      0
7        17    55     50     22
8              0      0      0

ACTL.  X    -25,500  |  T
       Y    +21,325  |  F  0
       Z   +150,000  |                M5/9
       C    +90,000  |
```

Fig. 4.4: Section of a tool table

The sequence of information in the illustrations at right is only one of many possibilities.

If a table is too large to show all information on one screen a ">>" or "<<" symbol appears in the line with the table name. By moving the cursor to the left or right with the arrow keys you can pan to the other columns.

You can save the tool length and tool radius as an actual position by pressing the ACTUAL POS. X, ACTUAL POS. Y or ACTUAL POS. Z soft key.

Abbreviation	Input	Dialog
T	Number by which the tool is called in the program	–
NAME	Name by which the tool is called in the program (up to 16 characters)	TOOL NAME ?
L	Compensation value for tool length	TOOL LENGTH L ?
R	Tool radius R	TOOL RADIUS R ?
DL	Delta value for tool length	TOOL LENGTH OVERSIZE ?
DR	Delta value for tool radius R	TOOL RADIUS OVERSIZE ?
TL	Tool lock	TOOL LOCKED ?
RT	Number of a Replacement Tool, if available (see also TIME2)	REPLACEMENT 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 reaches or exceeds this value, the TNC changes to the replacement 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 AGE ?
DOC	Comment on the tool (up to 16 characters)	TOOL DESCRIPTION ?
PLC	This function is not available yet.	PLC STATUS

Overview: Information in tool tables

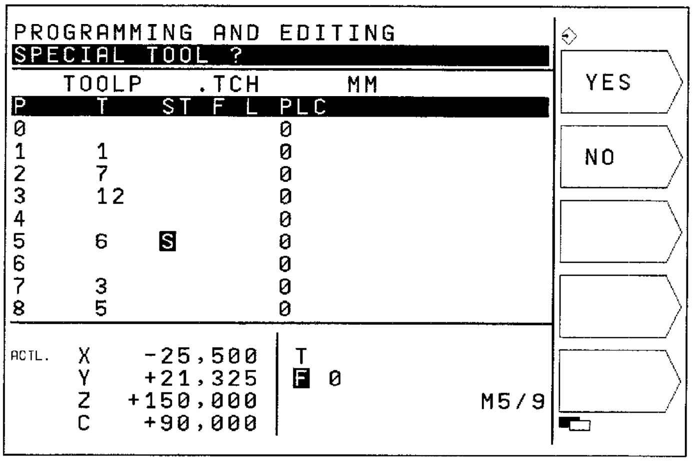
Abbreviation	Input	Dialog
CUT.	Number of cutting edges that are measured in automatic tool measurement (20 cutting edges maximum)	NUMBER OF TEETH ?
LTOL	Permissible deviation from tool length L for monitoring wear. If the entered value is exceeded, the TNC locks the tool (status L). Input range: 0 to 0.9999 mm 0: Wear monitoring is not active	WEAR TOLERANCE: LENGTH ?
RTOL	Permissible deviation from tool radius R for monitoring wear. If the entered value is exceeded, the TNC locks the tool (status L). Input range: 0 to 0.9999 mm 0: Wear monitoring is not active	WEAR TOLERANCE: RADIUS ?
DIRECT.	Cutting direction of the tool for dynamic tool measurement	CUTTING DIRECTION (M3 = –)
TT:R-OFFS	Offset between stylus center and tool center Default setting: Tool radius R	TOOL OFFSET: RADIUS ?
TT:L-OFFS	Tool end offset in addition to MP 6530 from top of stylus. Default setting: 0	TOOL OFFSET: LENGTH ?
LBREAK	Permissible deviation from tool length L for monitoring breakage. If the entered value is exceeded, the TNC locks the tool (status L). Input range: 0 to 0.9999 mm 0: Breakage monitoring is not active	BREAKAGE TOLERANCE: LENGTH ?
RBREAK	Permissible deviation from tool radius R for monitoring breakage. If the entered value is exceeded, the TNC locks the tool (status L). Input range: 0 to 0.9999 mm 0: Breakage monitoring is not active	BREAKAGE TOLERANCE: RADIUS ?

Overview: Information in tool tables for automatic tool measurement

Pocket table for tool changer

The table TOOLP.TCH is programmed for automatic tool changers.

In the pocket table the following information is assigned to the individual pocket numbers.



```
PROGRAMMING AND EDITING
SPECIAL TOOL ?
    TOOLP     .TCH       MM
P     T    ST  F  L  PLC
0                    0
1     1              0
2     7              0
3     12             0
4                    0
5     6     S        0
6                    0
7     3              0
8     5              0

ACTL.  X     -25,500  T
       Y     +21,325  F 0
       Z    +150,000                  M5/9
       C     +90,000
```

Fig. 4.5: Tool pocket table

Abbreviation	Input	Dialog
P	**P**ocket number of the tool in the magazine	–
T	**T**ool number	TOOL NUMBER
ST	If the tool is so large that it blocks the adjacent pockets in the tool magazine, you can lock the pockets before and after it in MP7264 (see page 11-5).	SPECIAL TOOL
F	Tool is always returned to a certain pocket in the magazine	FIXED POCKET
L	Tool pocket is locked	POCKET LOCKED
PLC	This function is not available yet	PLC STATUS

Overview: Information in tool pocket tables

Calling tool data

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

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

To call tool data:

T ▶ **TOOL NUMBER ?**

e.g. 5	Enter the number of the tool as defined in the tool table or in a G99 block, for example 5.
G 1 7	Select the spindle axis Z.
S 5 0 0 END □	Enter the spindle speed, e.g. S = 500 rpm.

Resulting NC block: T5 G17 S500

Tool pre-selection with tool tables

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

Tool change

Automatic tool change

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

If your machine is built for automatic tool changing, 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 3-4)
- Change the tool
- Continue the program run (see page 3-5)

Tool change position

A tool change position must lie next to or above the workpiece to prevent tool collision. With the miscellaneous functions M91 and M92 (see page 5-38) 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 spindle to an uncompensated position.

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

4.3 Tool Compensation Values

For each tool, the control adjusts the spindle path in the tool axis by the compensation value for the tool length. In the working plane it compensates the tool radius.

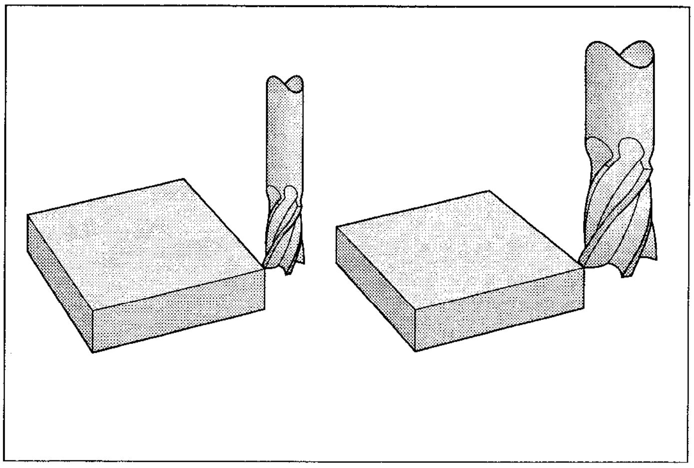


Fig. 4.6: The TNC must compensate the length and radius of the tool

Effect of tool compensation values

Tool length

The compensation value for the tool length is calculated as follows:

Compensation value = L + DL_TAB

where	L:	is the tool length L (from the G99 block or the tool table)
	DL_TAB:	is the oversize for length DL in the tool table

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

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

If a positive length compensation was in effect before the tool call T0, the distance to the workpiece is decreased. In an incremental movement of the tool axis immediately after a tool call with T, the difference in length between the old and new tools is moved in addition to the programmed value.

Tool radius

The compensation value for the tool radius is calculated as follows:

Compensation value = R + DR_TAB

where	R:	is the tool radius R (from the G99 block or the tool table)
	DR_TAB:	is the oversize for radius DR in the tool table

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

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

Tool radius compensation

Tool traverse can be programmed:

- Without radius compensation: G40
- With radius compensation: G41 or G42
- As single-axis movements with G43 or G44.

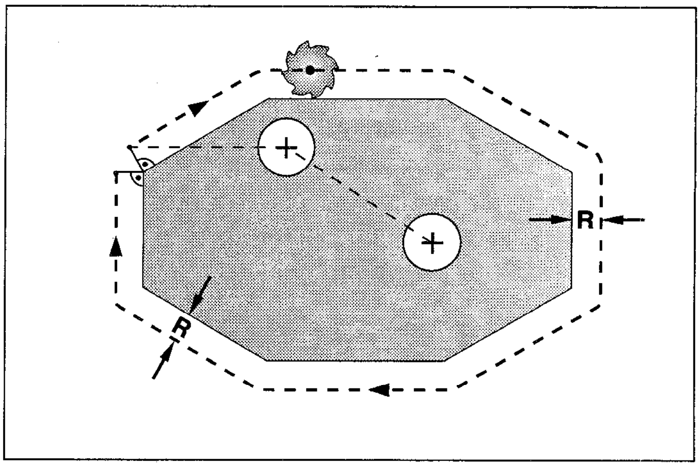


Fig. 4.7: Programmed contour (—, +) and the path of the tool center (- - -)

Traverse without radius compensation: G40

The tool center moves to the programmed coordinates.

Applications:

- Drilling and boring
- Pre-positioning

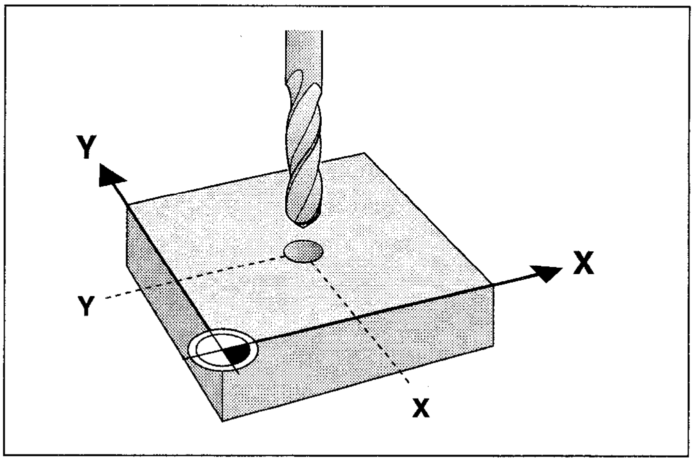


Fig. 4.8: These drilling positions are entered without radius compensation

Traverse with radius compensation G41, G42

The tool center moves to the left (G41) or to the right (G42) of the programmed contour at a distance equal to the tool radius. "Right" or "left" is meant as seen in the direction of tool movement as if the workpiece were stationary.

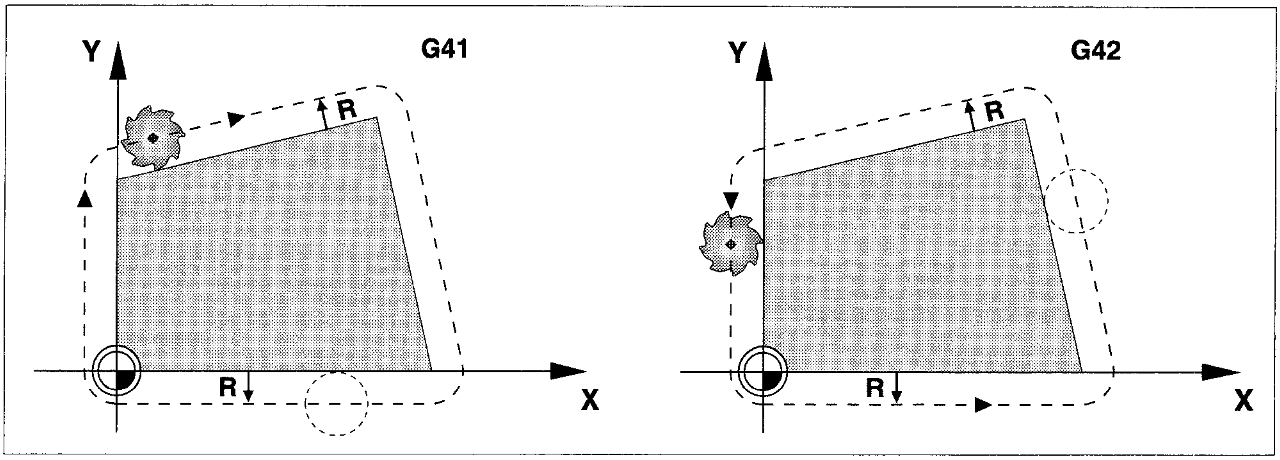


Fig. 4.9: The tool moves at left (G41) or at right (G42) of the workpiece during milling

Between two program blocks with differing radius compensation you must program at least one block without radius compensation (that is, with G40). Radius compensation is not in 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 possible only for single-axis movements in the working plane: The programmed tool path is shortened (G44) or lengthened (G43) by the tool radius.

Applications:

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

- G43 and G44 are available whenever a positioning block is created with only one axis.
- The machine tool builder can set a machine parameter to inhibit such single-axis positioning blocks.

Machining corners

If you work without radius compensation, you can influence the machining of outside corners with the miscellaneous function M90 (see page 5-35).

Outside corners

The control 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 machine strain, for example for very sharp changes in direction.

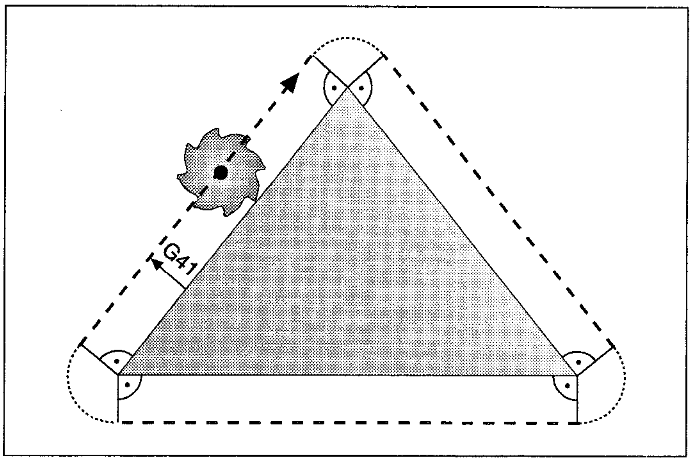


Fig. 4.10: The tool "rolls around" outside corners

Inside corners

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

The control 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 at inside corners.

When two or more inside corners adjoin, the chosen tool radius must be small enough to fit in the programmed contour.

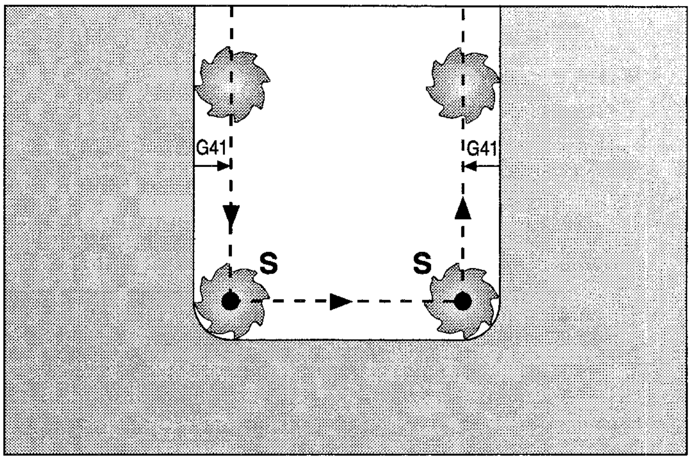


Fig. 4.11: Tool path for inside corners

4.4 Program Creation

To create a new part program

PGM MGT	Call the file management.

Select any program.

PROGRAM NUMBER=

e.g. 7 4 3 ENT	Enter the name of the new program, for example 743.I.

PROGRAM INPUT: ISO / MM

CHANGE HDH/DIN	Create a program in ISO format or a HEIDENHAIN dialog program. Either DIN or HDH appears in the dialog line.
CHANGE MM/INCH	Select millimeters or inches as units of measure. Either MM or INCH appears in the dialog line.

ENT	Confirm your selection.

Defining the blank form

If you wish to use the 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.

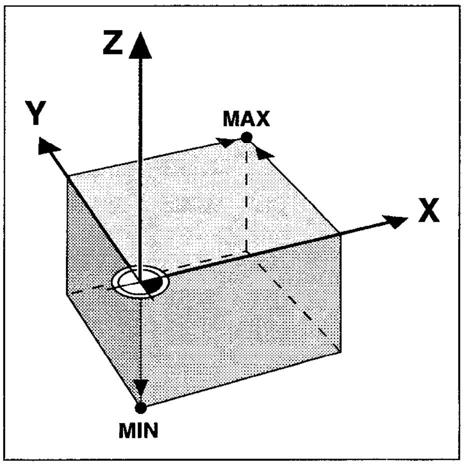


Fig. 4.12: The MIN and MAX points define the blank form

The ratio of the blank-form side lengths must be less than 84:1.

MIN and MAX points

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

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

G 3 0	G function for entering the MIN point.
G 1 7	Select the tool axis: G17 means Z axis.
e.g. X 0 Y 0 Z 4 0 – END	Enter the X, Y and Z coordinates of the MIN point in sequence, close the block with END.

G 3 1	G function for entering the MAX point.
G 9 0	Input of absolute values, or
G 9 1	incremental values.
e.g. X 1 0 0 Y 1 0 0 Z 0 END	Enter the X, Y and Z coordinates of the MAX point in sequence, close the block with END.

The entered program section appears on the TNC screen:

```
% 743 G71 *
```

Block 1: Program beginning, name, unit of measure.

```
N10 G30 G17 X+0 Y+0 Z-40 *
```

Block 2: Spindle axis, MIN point coordinates.

```
N20 G31 G90 X+100 Y+100 Z+0 *
```

Block 3: MAX point coordinates.

```
N9999 % 743 G71 *
```

Block 4: Program end, name, unit of measure.

The unit of measure used in the program appears behind the program name (G71 = mm).

4.5 Entering Tool-Related Data

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

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

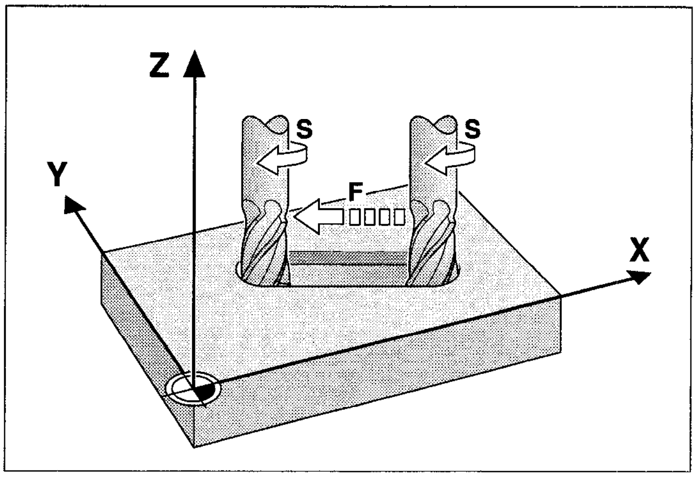


Fig. 4.13: Feed rate F and spindle speed S of the tool

Feed rate F

The feed rate is the speed in mm/min (or inch/min) with which the tool center moves.

Input range:
F = 0 to 99 999 mm/min (3937 inch/min)

The maximum feed rate is set in machine parameters individually for each axis.

To set the feed rate:

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

Rapid traverse

You can program rapid traverse directly with the G00 function.

Duration of feed rate F

A feed rate that is entered as a numerical value remains in effect until the control executes a block in which another feed rate has been programmed.

If the new feed rate is G00 (rapid traverse), the feed rate will return to the last numerically entered feed rate as soon as the next block with G01 is executed.

Changing the feed rate F

You can vary the feed rate by turning the knob for feed rate override on the operating panel (see page 2-6).

Spindle speed S

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

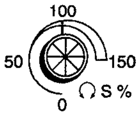
Input range:
S = 0 to 99 999 rpm

To change the spindle speed S in the part program:

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

Resulting NC block: T1 G17 S1000

To change the spindle speed S during program run:

	You can vary the spindle speed S on machines with stepless ballscrew drives by turning the override knob on the operating panel.

4.6 Entering Miscellaneous Functions and STOP

The machine tool builder determines which M functions are available on your TNC and what effects they have. Your machine manual provides more information on this subject.

The M functions (M for miscellaneous) affect:

- Program run
- Machine functions
- Tool behavior

On the inside back cover of this manual you will find a list of M functions that are predetermined for the TNC. The list indicates whether an M function begins at the start or at the end of the block in which it is programmed.

You can program more than one M function in an NC block as long as these M functions are independent of each other. The list of M functions on the inside back cover shows the different groups of M functions.

A program run or test run is interrupted when it reaches an NC block containing the function G38.

If you wish to interrupt the program run or test run for a certain duration, use the cycle G04: DWELL TIME (see page 8-38).

4.7 Actual Position Capture

Sometimes you may want to enter the actual position of the tool in a specific 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. This feature can be used, for example, to enter the tool length.

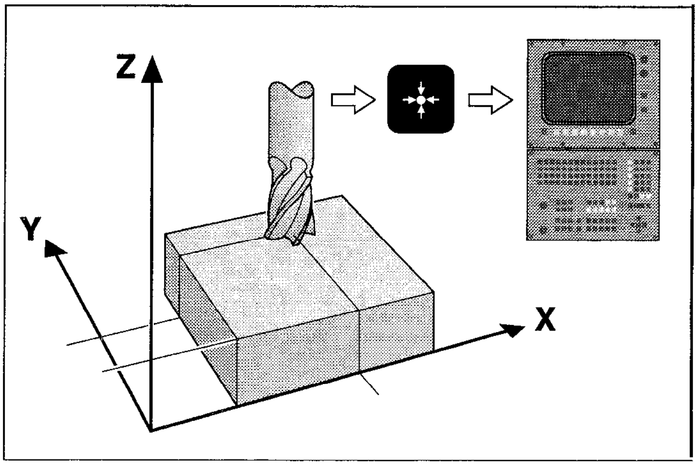


Fig. 4.14: Storing the actual position in the TNC

To capture the actual position:

MANUAL OPERATION

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

PROGRAMMING AND EDITING

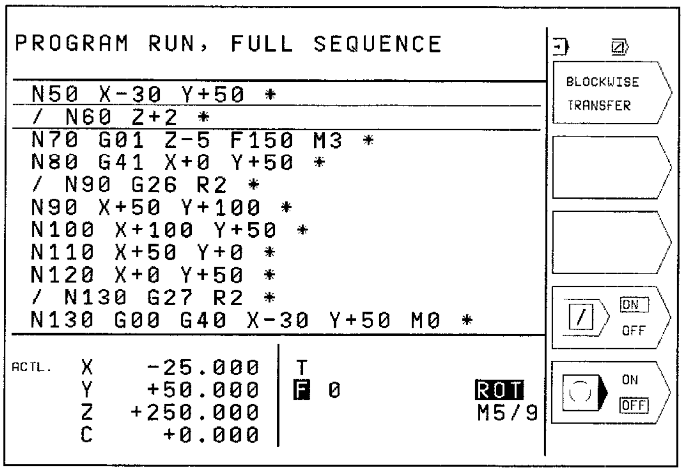
Select or create the program block in which you wish to enter the actual position of the tool.

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

During a program run or test run, the TNC ignores the program blocks that are marked to be jumped over (see page 3-7).



```
PROGRAM RUN, FULL SEQUENCE
 N50 X-30 Y+50 *
 / N60 Z+2 *
 N70 G01 Z-5 F150 M3 *
 N80 G41 X+0 Y+50 *
 / N90 G26 R2 *
 N90 X+50 Y+100 *
 N100 X+100 Y+50 *
 N110 X+50 Y+0 *
 N120 X+0 Y+50 *
 / N130 G27 R2 *
 N130 G00 G40 X-30 Y+50 M0 *
ACTL.  X     -25.000  T
       Y     +50.000  F 0          ROT
       Z    +250.000               M5/9
       C      +0.000
```

Fig. 4.15: Blocks marked with "/"

To mark blocks

Select a block that should not be executed each time.

/	Mark the block by entering the "/" sign at the beginning of the block.

Blocks containing a tool definition with G99 must not be skipped.

To erase the "/" sign

Select the block in which you wish to erase the "/" sign.

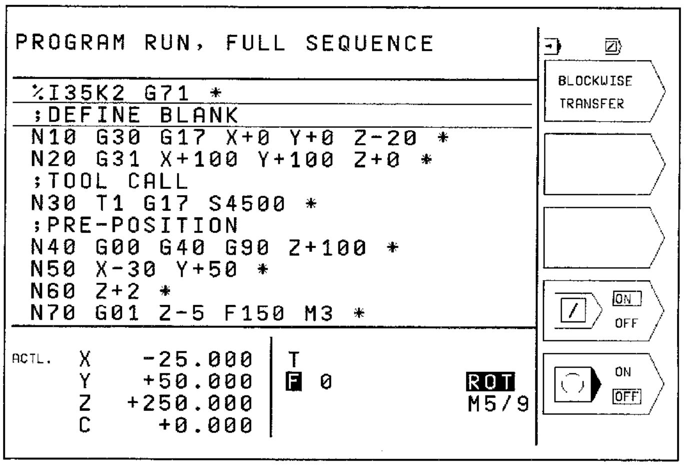
⌫	Erase the "/" sign.

4.9 Entering Comments in the Part Program

In the PROGRAMMING AND EDITING mode of operation you can add comments to the part program. They can be entered as separate blocks or at the end of blocks.

Applications:

- Describing programming steps
- Adding general information



```
PROGRAM RUN, FULL SEQUENCE

 %I35K2 G71 *
 ;DEFINE BLANK
 N10 G30 G17 X+0 Y+0 Z-20 *
 N20 G31 X+100 Y+100 Z+0 *
 ;TOOL CALL
 N30 T1 G17 S4500 *
 ;PRE-POSITION
 N40 G00 G40 G90 Z+100 *
 N50 X-30 Y+50 *
 N60 Z+2 *
 N70 G01 Z-5 F150 M3 *

ACTL. X    -25.000 | T
      Y    +50.000 | F 0        ROT
      Z   +250.000 |            M5/9
      C     +0.000 |
```

Fig. 4.16: Adding comments to the part program

To enter comments

; Enter a semicolon.

Enter the comment by using the alphabetic and numerical keys.

ENT Terminate the comment.

5 Programming Tool Movements

5.1 General Information on Programming Tool Movements

You always program tool movements 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).

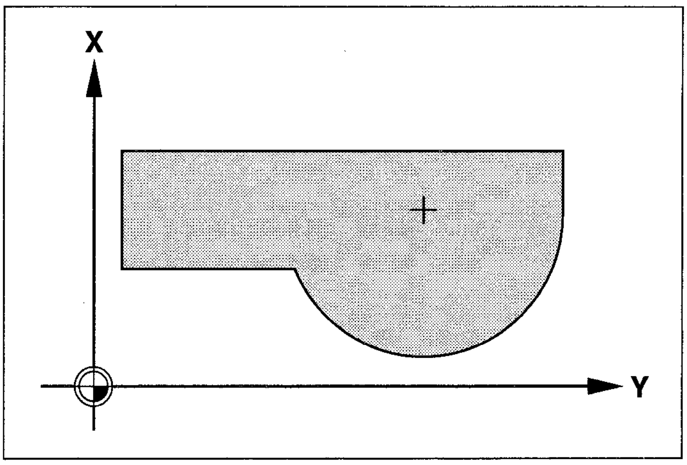


Fig. 5.1: A contour consists of straight lines and circular arcs

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

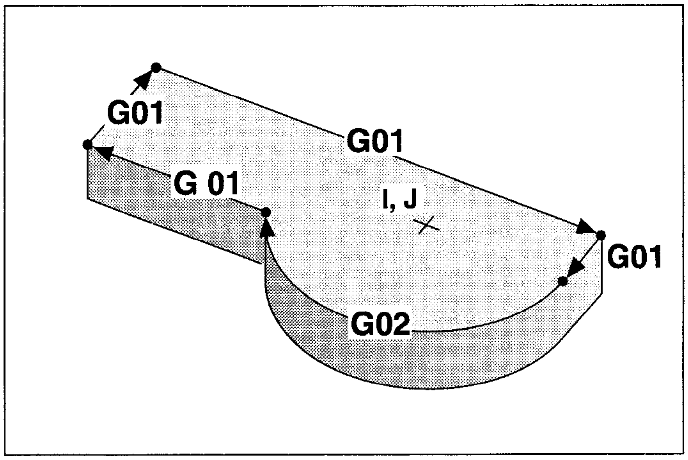


Fig. 5.2: Contour elements are programmed and executed in sequence

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 variables 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

and others.

5.2 Contour Approach and Departure

A convenient way to approach or depart the workpiece is on an arc that 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. 5.3, 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.

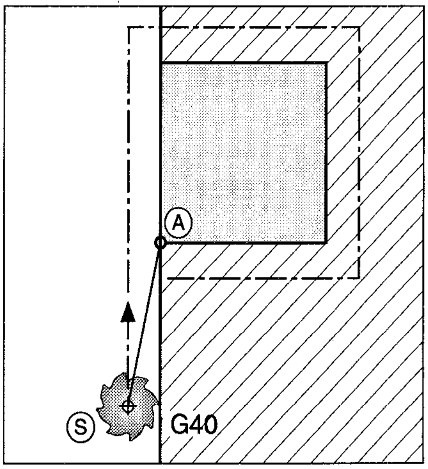


Fig. 5.3 : Starting point Ⓢ of machining

First contour point

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

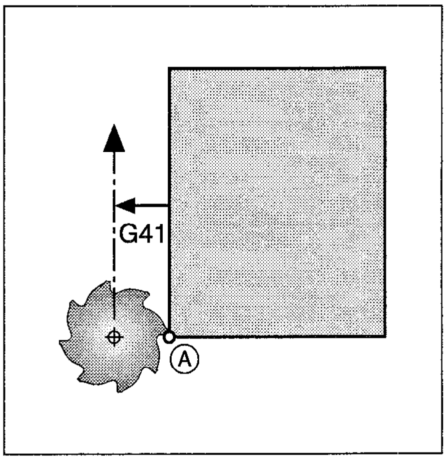


Fig. 5.4 : First contour point for machining

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
```

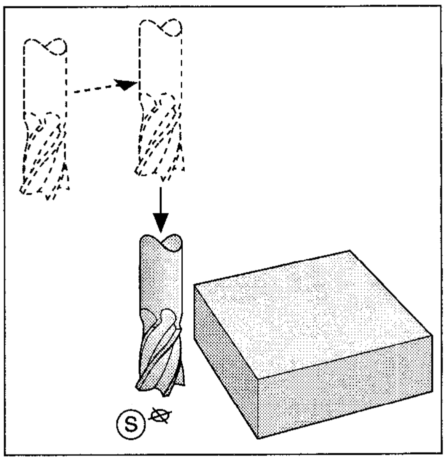


Fig. 5.5 : Separate movement of the spindle when there is danger of collision

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.

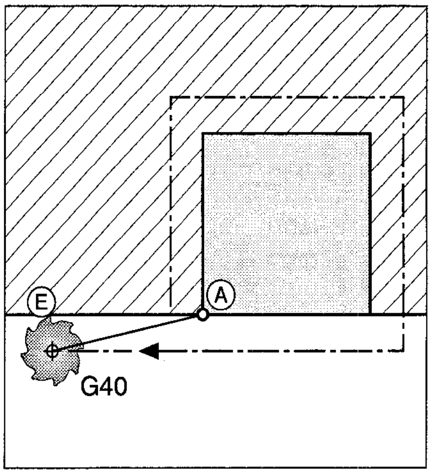


Fig. 5.6 : End point Ⓔ for machining

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

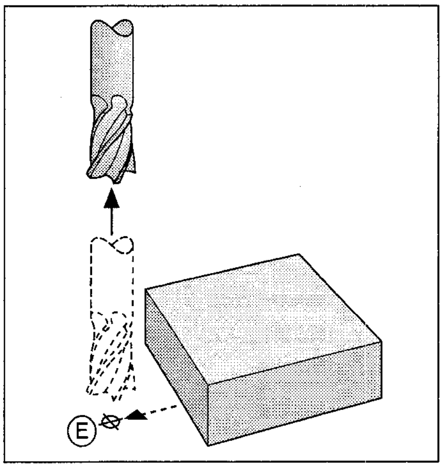


Fig. 5.7 : Retract spindle axis separately

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 (SE).

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.

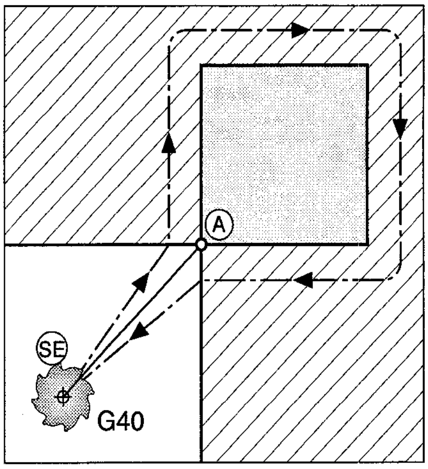


Fig. 5.8: Common starting and end point

Tangential approach and departure

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

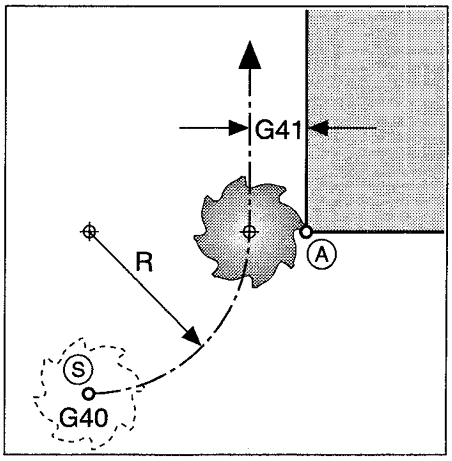


Fig. 5.9: Soft contour approach

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).

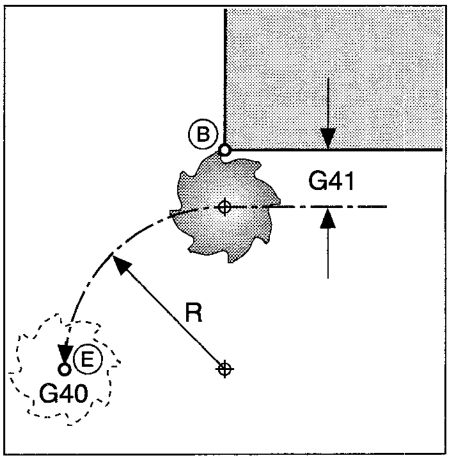


Fig. 5.10: Soft contour departure

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. Only *one* axis is programmed in the block.

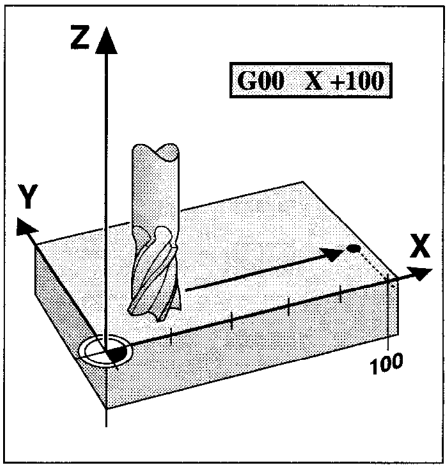


Fig. 5.11: Paraxial movement

Movement in the main planes

The tool moves to the programmed position on a straight line or circular arc in a plane. *Two* axes are programmed in the block.

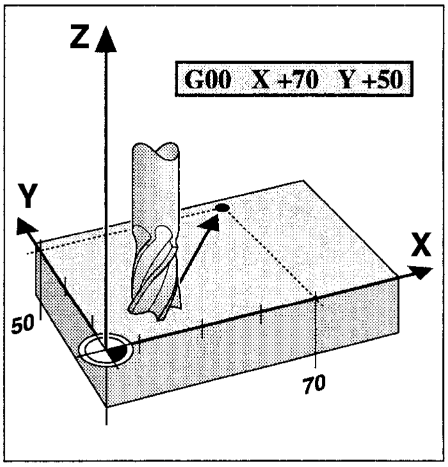


Fig. 5.12: Movement in a main plane (XY)

Movement of three machine axes (3D movement)

The tool moves in a straight line to a position programmed in *three* axes.

Exception: A helical path is created by combining a circular movement in a plane with a linear movement perpendicular to the plane.

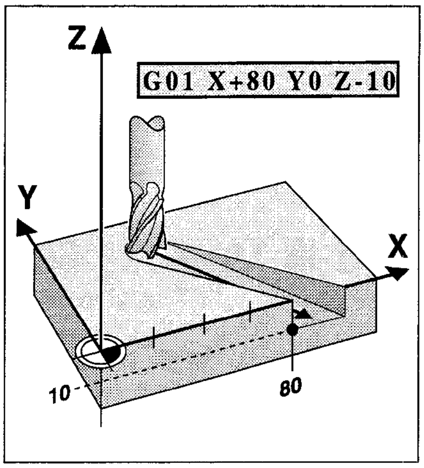


Fig. 5.13: Three-dimensional movement

Overview of path functions

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

5.4 Path Contours – Cartesian Coordinates

G00: Straight line with rapid traverse

G01: Straight line with feed rate F ...

To program a straight line, you enter:

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

The tool moves in a straight line from its current position to the end point Ⓔ. The starting position Ⓢ is approached in the preceding block.

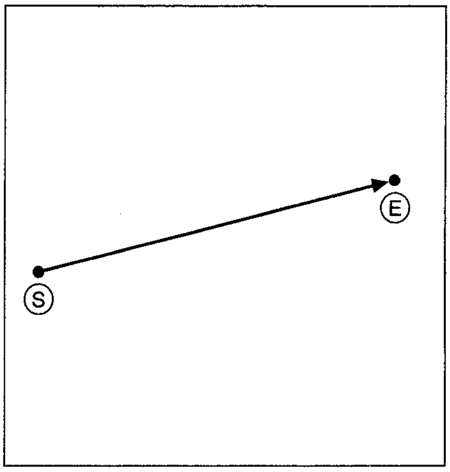


Fig. 5.14: Linear movement

To program a straight line:

G 0 0	G function for straight line with rapid traverse.
If necessary G 9 1	Specify as relative coordinate, for example G91 X–50 mm.
X	Select the axis (typewriter keyboard), for example X.
5 0	Enter the coordinate of the end point.
If necessary –	For negative coordinates, press the minus key once, e.g. X = –50 mm.
Y ⋮ Z	Enter all further coordinates of the end point.

⋮

⋮

G 4 1	The TNC must move the tool by its radius to the left of the programmed contour.
G 4 2	The TNC must move the tool by its radius to the 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 you have entered all the coordinates, conclude the block with END.

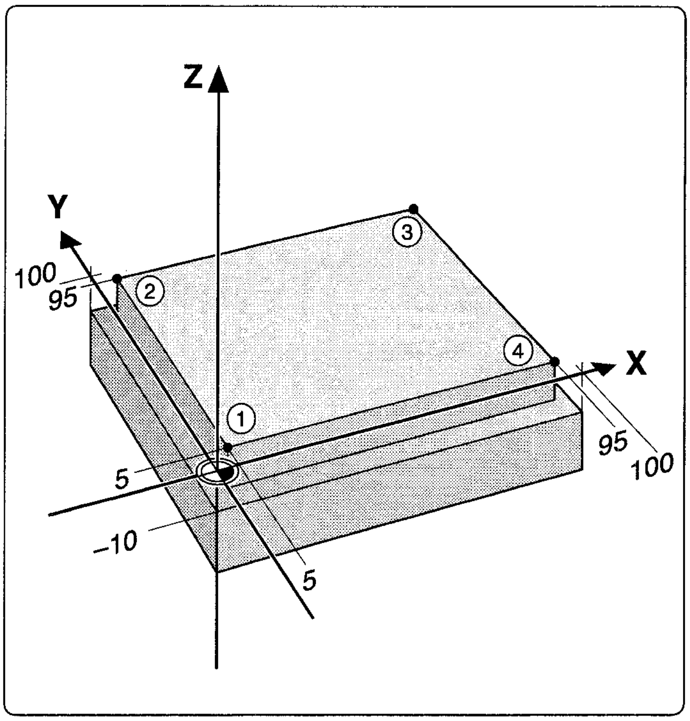
*Resulting NC block: N25 G00 G42 G91 X+50 G90 Y+10 Z–20 M3 **

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.

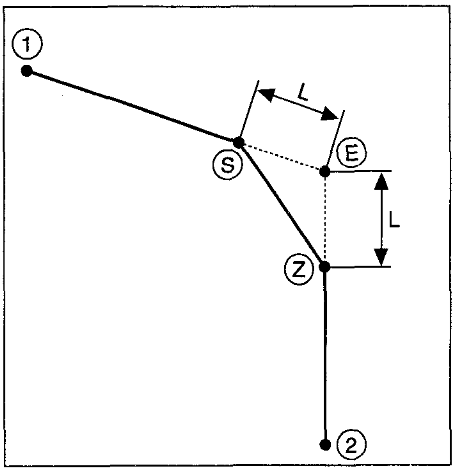


Fig. 5.15: Chamfer from Ⓢ to Ⓩ

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

Prerequisites

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

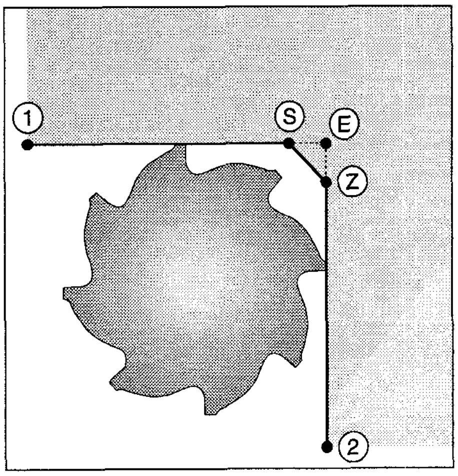


Fig. 5.16: Tool radius too large

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

To program a chamfer:

G 2 4 ENT	Select the chamfer function.
CHAMFER SIDE LENGTH ?	
5 END	Enter the length to be removed from each side of the corner, for example 5 mm.

*Resulting NC block: G24 R5**

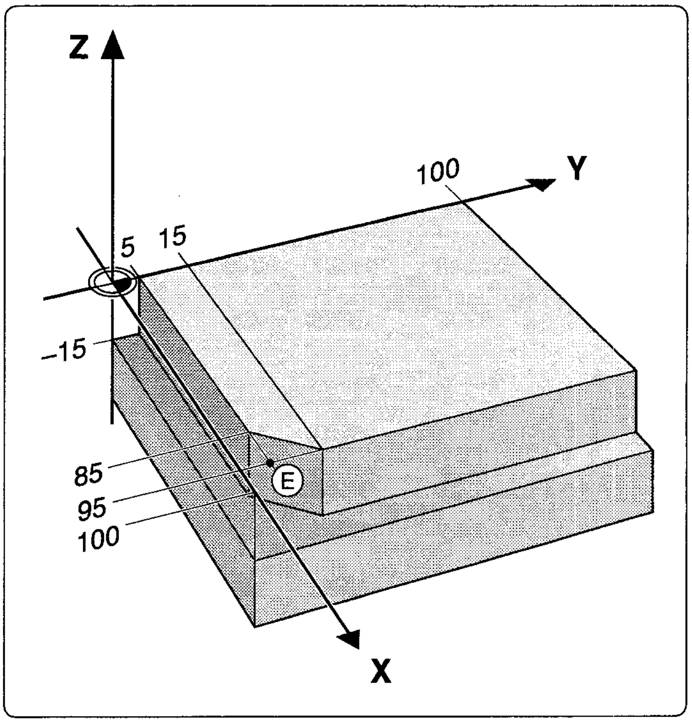
Example for exercise: Chamfering a corner

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

Length of chamfer: CHF = 10 mm

Tool radius: R = +10 mm

Milling depth: Z = –15 mm



Part program

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

Circles and circular arcs

Here the TNC moves two axes simultaneously in a circular path relative to the workpiece.

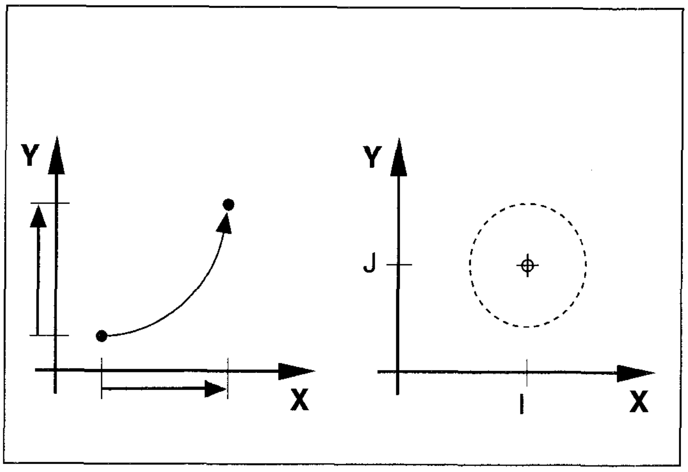


Fig. 5.17: Circular arc and circle center

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.

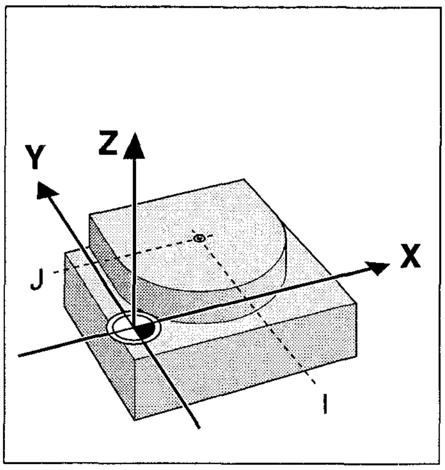


Fig. 5.18: Circle center coordinates

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

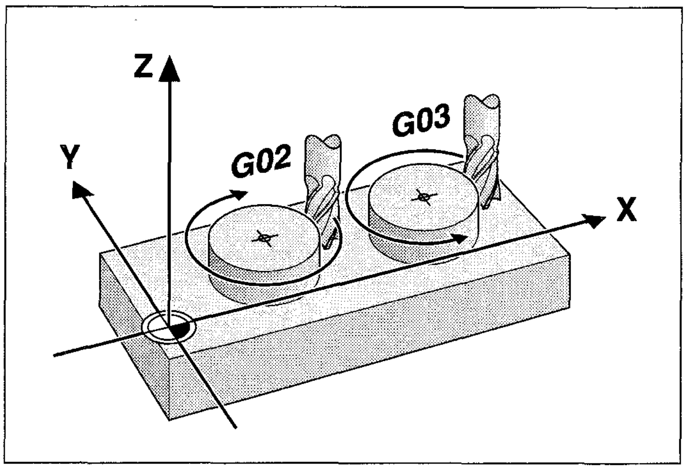


Fig. 5.19: Direction of rotation for circular movement

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. 5.20: 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.

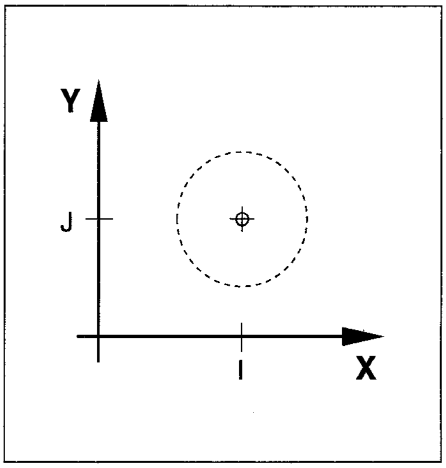


Fig. 5.21: Circle center I, J

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.

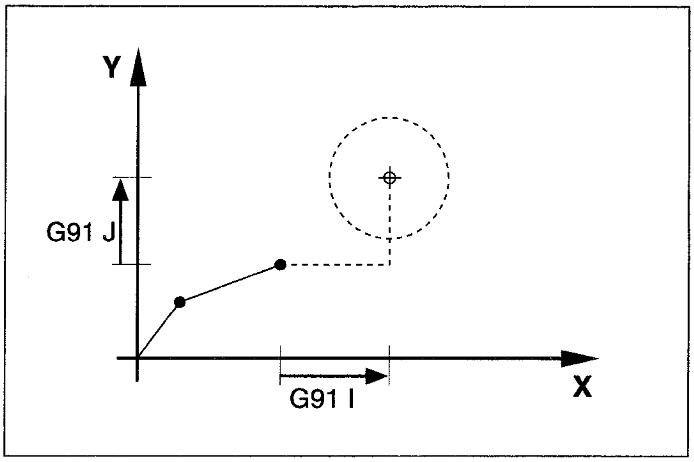


Fig. 5.22: Incremental coordinates for a circle center

- 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

Prerequisites

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

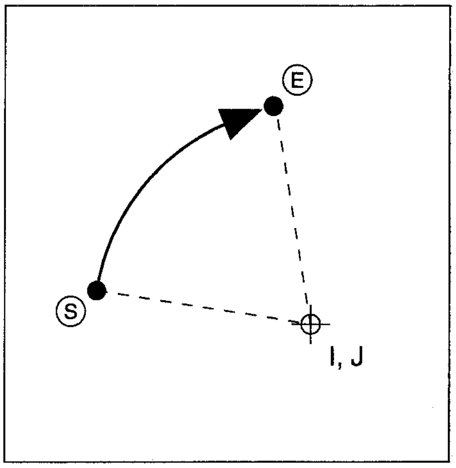


Fig. 5.23: Circular path from Ⓢ to Ⓔ around I, J

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.

- For a full circle, the end point in the G02/G03 block should be the same as the starting point.

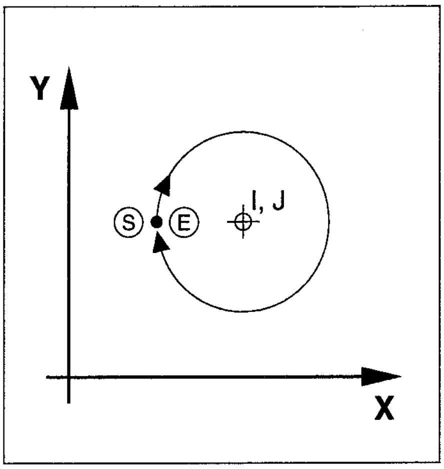


Fig. 5.24: Full circle around I, J with a G02 block

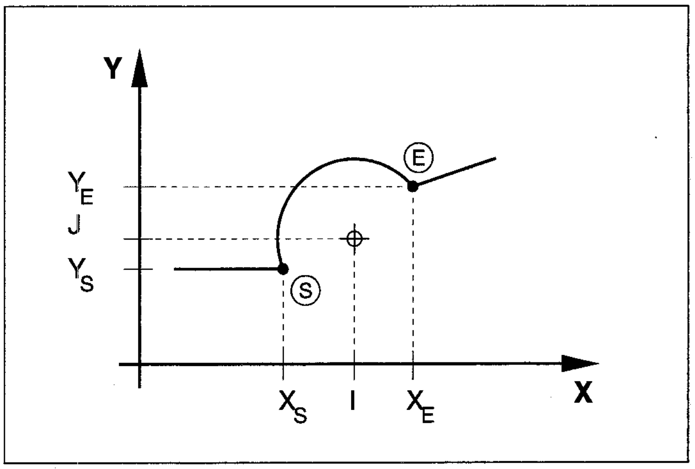


Fig. 5.25: Coordinates of an arc

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.

If necessary, enter also:

- Radius compensation
- Feed rate
- Miscellaneous function

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

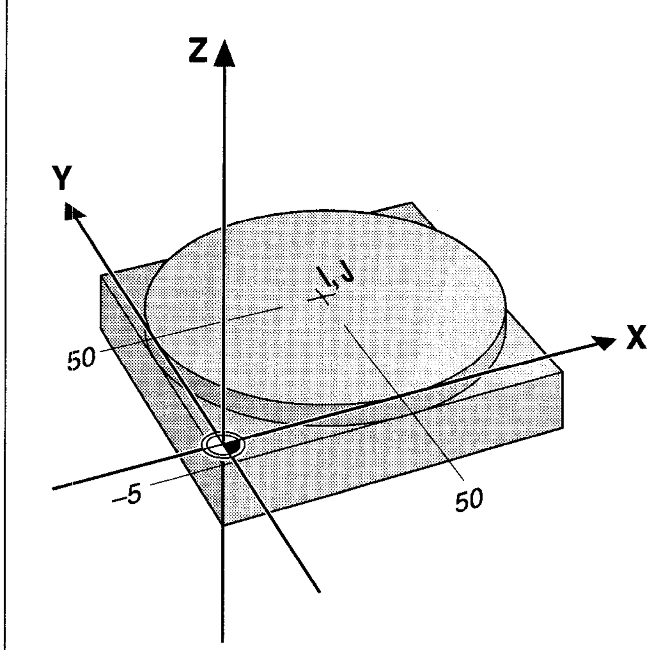
Example for exercise: Mill a full circle with one block

Circle center: I = 50 mm
J = 50 mm

Beginning and end
of the arc: X = 50 mm
Y = 0 mm

Milling depth: Z = –5 mm

Tool radius: R = 15 mm



Part program

%S520I G71 *	Begin the program
N10 G30 G17 X+1 Y+1 Z–20 *	Workpiece blank MIN point
N20 G31 G90 X+100 Y+100 Z+0 *	Workpiece blank MAX point
N30 G99 T6 L+0 R+15 *	Define the tool
N40 T6 G17 S1500 *	Call the tool
N50 G00 G40 G90 Z+100 M06 *	Retract and insert tool
N60 X+50 Y–40 *	Pre-position in the working plane
N70 Z-5 M03 *	Move tool to working depth
N80 I+50 J+50 *	Coordinates of the circle center
N90 G01 G41 X+50 Y+0 F100 *	Approach first contour point with radius compensation at machining feed rate
N100 G26 R10 *	Soft (tangential) approach
N110 G02 X+50 Y+0 *	Mill arc around circle center I,J; direction of rotation negative (clockwise); coordinates of end point X = +50 mm, Y = +0
N120 G27 R10 *	Soft (tangential) departure
N130 G00 G40 X+50 Y–40 *	Depart the contour, cancel radius compensation
N140 Z+100 M02 *	Retract in the infeed axis
N99999 %S520I G71 *	

G02/G03/G05: Circular path with defined radius

The tool moves on a circular path with radius R.

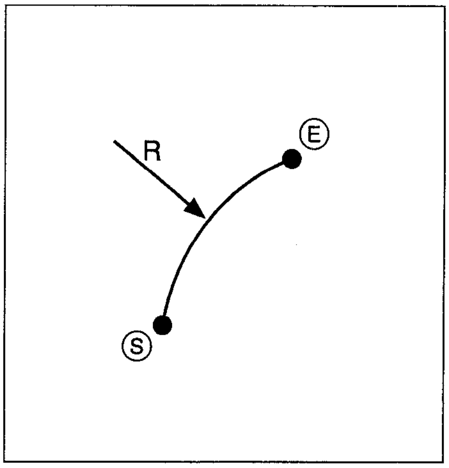


Fig. 5.26: Circular path from Ⓢ to Ⓔ with radius R

Defining the direction of rotation

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

Input

- Coordinates of the end point of the arc
- Radius R of the arc

- 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.

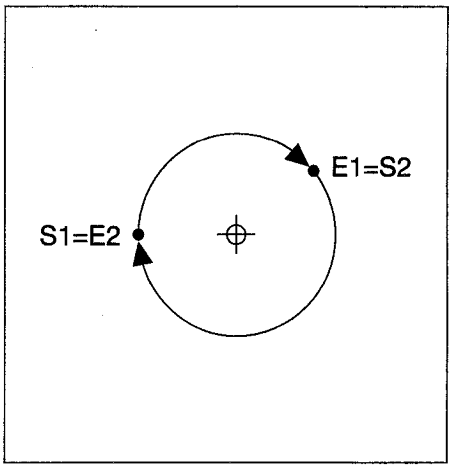


Fig. 5.27: Full circle with two G02 blocks

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 differ in their lengths and curvatures.

An arc larger than a semicircle: CCA>180°
Input: Radius R with negative sign (R<0).

An arc smaller than a semicircle: CCA<180°
Input: Radius R with positive sign (R>0).

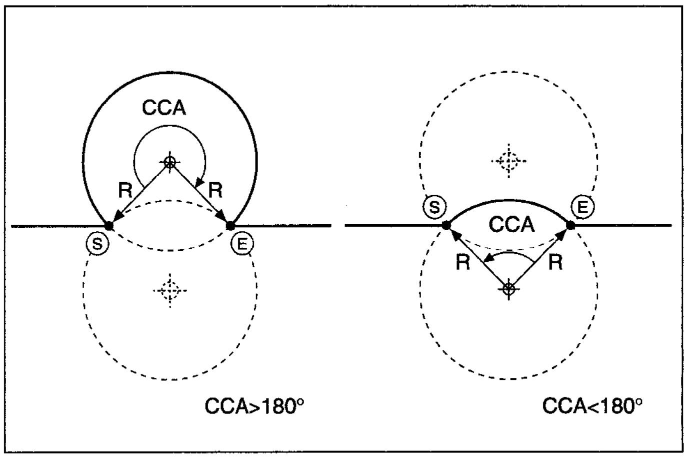


Fig. 5.28: Arcs with central angles greater than and less than 180°

Contour curvature and direction of rotation

The direction of rotation determines the type of arc:

- Convex (curving outward), or

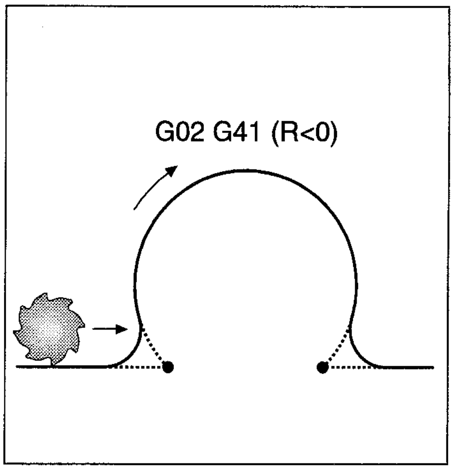


Fig. 5.29: Convex path

- Concave (curving inward)

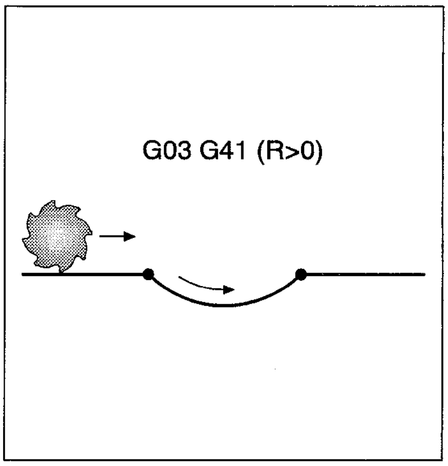


Fig. 5.30: Concave path

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).

If necessary, enter also:

- Radius compensation
- Feed rate
- Miscellaneous function

Resulting NC block: G02 G41 X+10 Y+2 R–5

Example for exercise: Milling a concave semicircle

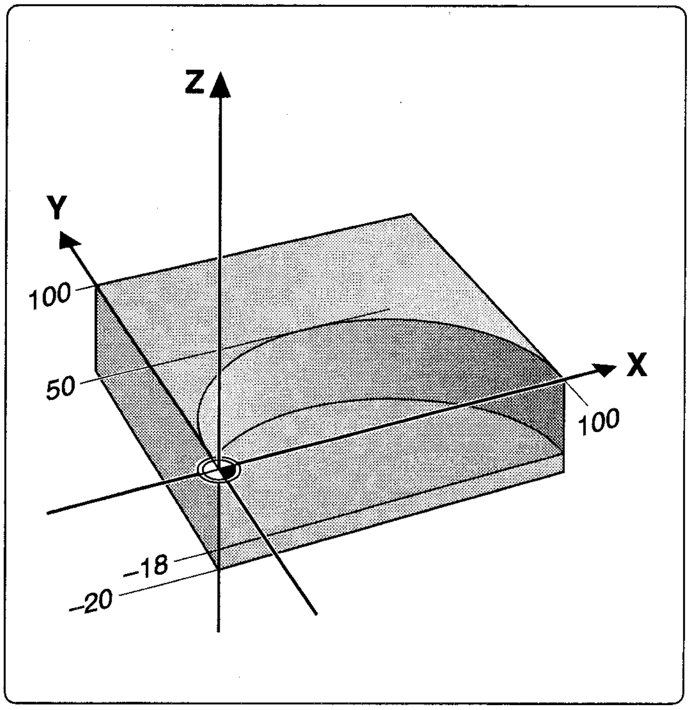
Semicircle radius: R = 50 mm

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 = 100 mm, Y = 0;
                                                        radius = 50 mm, 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 that tangentially connects to the arc.

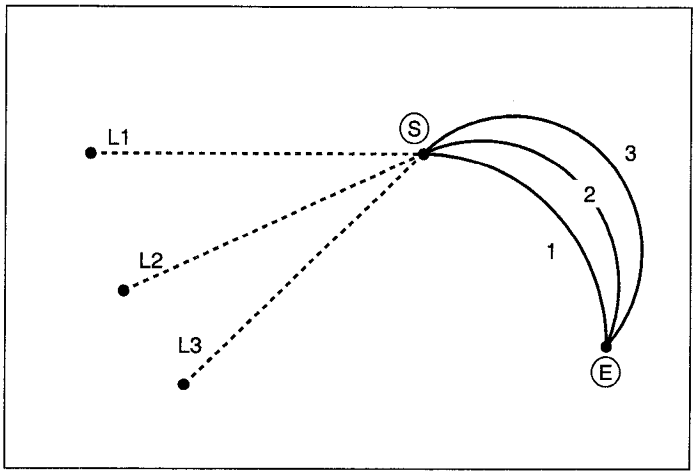


Fig. 5.31: The straight line ① - ② is connected tangentially to the circular arc Ⓢ - Ⓔ

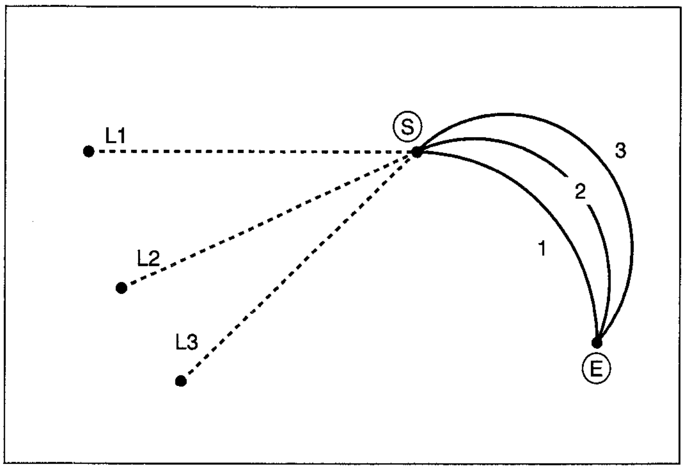


Fig. 5.32: The path of a tangential arc depends on the preceding contour element

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.

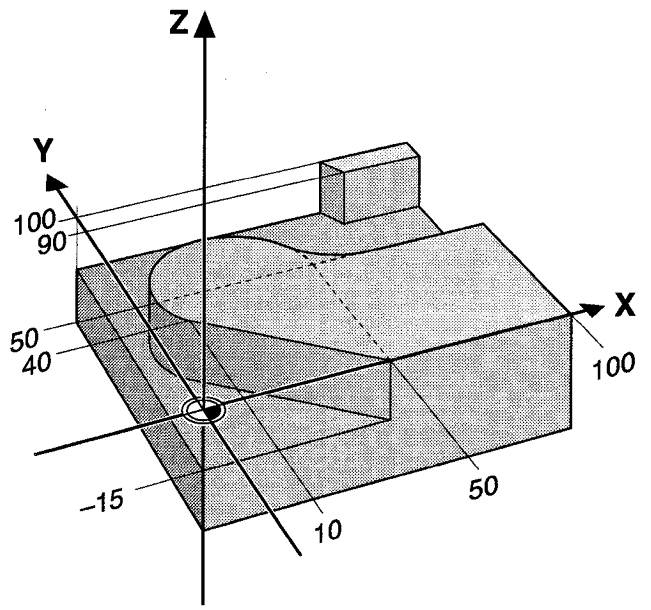
If necessary, enter also:

- 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.

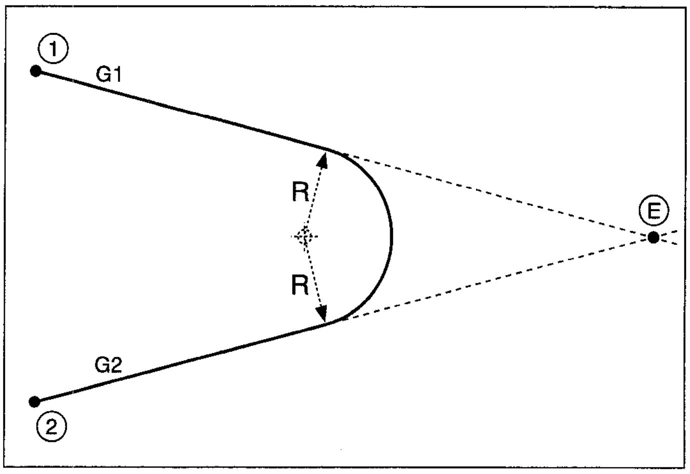


Fig. 5.33: Rounding radius R between G1 and G2

- 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.

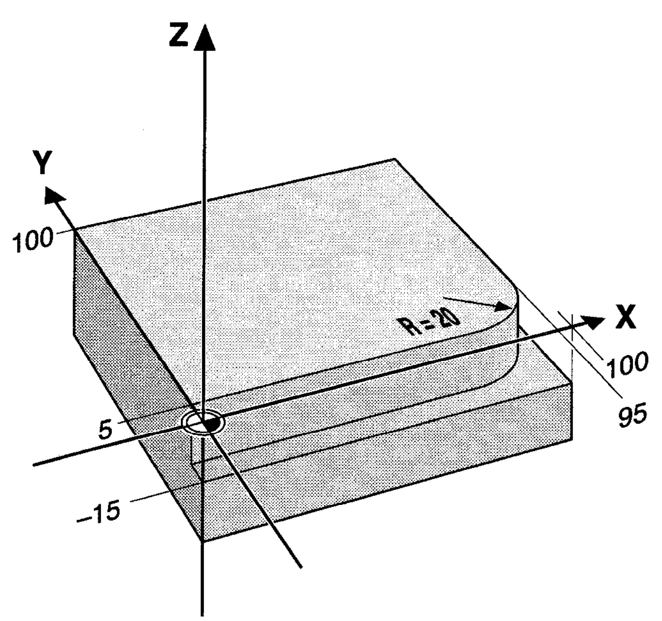
To program a tangential arc between two contour elements:

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 for:

- Positions on circular arcs
- Workpiece drawing dimensions in degrees

Polar coordinates are explained in detail in the section "Fundamentals of NC" (page 1-16).

Polar coordinate origin: Pole I, J, K

The pole can be defined anywhere in the program before blocks containing polar coordinates. Like 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
X Y	I, J
Y Z	J, K
Z X	K, I

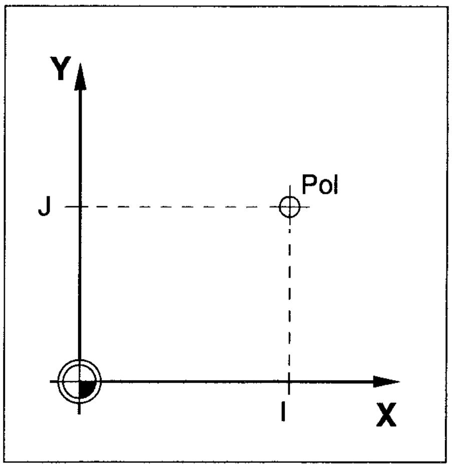


Fig. 5.34: The pole is the same as a circle center

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

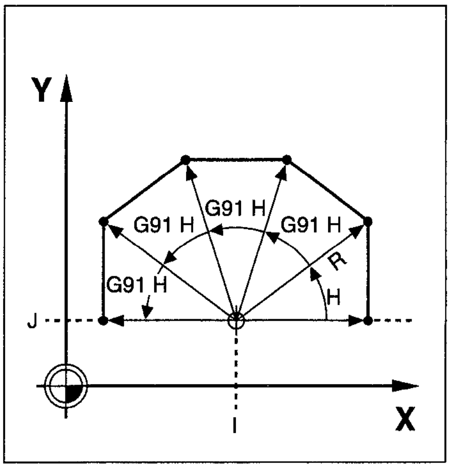


Fig. 5.35: Contour consisting of straight lines with polar coordinates

G 1 0	Straight line in polar coordinates with rapid traverse
R 5	Enter radius R from pole to end point of line (here, R = 5 mm).
H 3 0 END □	Enter angle H from angle reference axis to R (here, H = 30°).

*Resulting NC block: G10 R5 H30 **

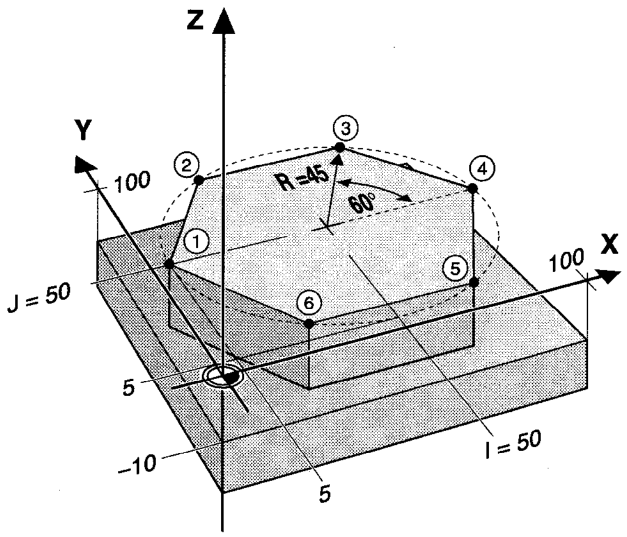
Example for exercise: Milling a hexagon

Corner point coordinates:

①	H = 180°	R = 45 mm
②	H = 120°	R = 45 mm
③	H = 60°	R = 45 mm
④	H = 0°	R = 45 mm
⑤	H = 300°	R = 45 mm
⑥	H = 240°	R = 45 mm

Milling depth: Z = –10 mm

Tool radius: R = 5 mm



Part program

```
%S530I G71 * ........................................ Begin program
N10 G30 G17 X+0 Y+0 Z–20 * .......................... Define the workpiece blank
N20 G31 G90 X+100 Y+100 Z+0 *
N30 G99 T1 L+0 R+17 * ............................... Define the tool
N40 T1 G17 S3200 * .................................. Call the tool
N50 G00 G40 G90 Z+100 M06 * ......................... Retract and insert tool
N60 I+50 J+50 * ..................................... Set pole
N70 G10 R+70 H–190 * ................................ Pre-position in the working plane with polar
                                                      coordinates
N80 Z–10 M03 * ...................................... Move tool to working depth
N90 G11 G41 R+45 H+180 F100 * ....................... Move to contour point 1
N100 H+120 * ........................................ Move to contour point 2
N110 H+60 * ......................................... Move to contour point 3
N120 G91 H–60 * ..................................... Move to contour point 4, incremental dimensions
N130 G90 H–60 * ..................................... Move to contour point 5, absolute dimensions
N140 H+240 * ........................................ Move to contour point 6
N150 H+180 * ........................................ Move to contour point 1
N160 G10 G40 R+70 H+170 * ........................... Depart contour, cancel radius compensation
N170 Z+100 M02 * .................................... Retract in the infeed axis
N99999 %S530I G71 *
```

G12/G13/G15: Circular path around pole I, J, K

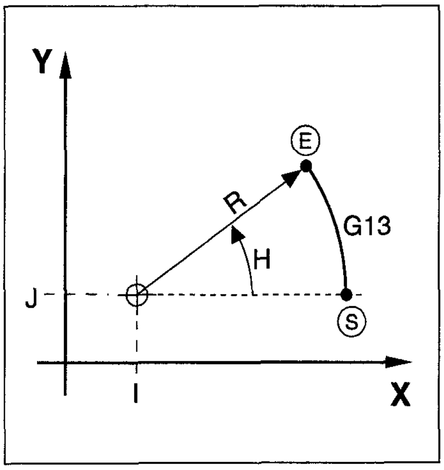


Fig. 5.36: Circular path around a pole

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 (the last programmed direction of rotation is used)	G15

G 1 2	Circle, polar coordinates, clockwise
H 3 0 END	Enter angle H for the end point of the arc (here, H = 30°). Conclude the block.

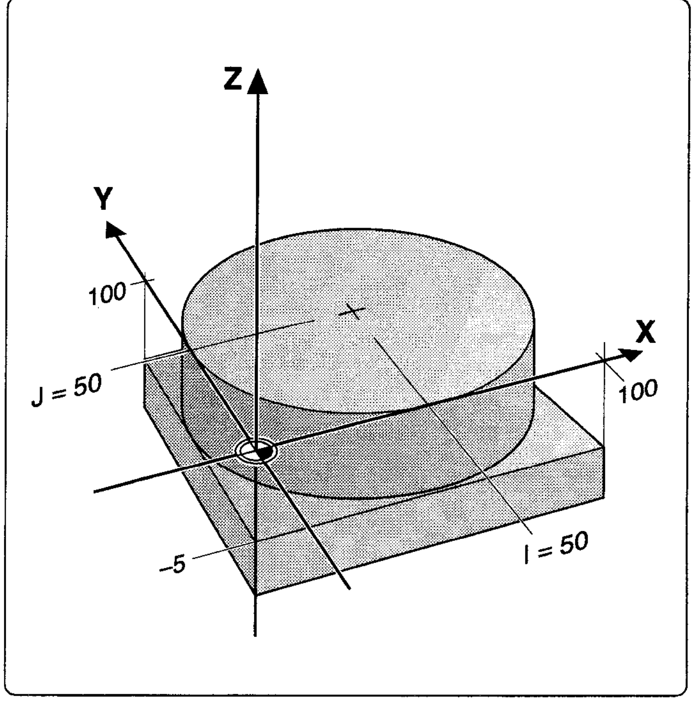
If necessary, enter also:

Radius compensation R
Feed rate F
Miscellaneous function M

*Resulting NC block: G12 H30 **

Example for exercise: Milling a full circle

Circle center coordinates:	X	=	50 mm
	Y	=	50 mm
Radius:	R	=	50 mm
Milling depth:	Z	=	– 5 mm
Tool radius:	R	=	15 mm



Part program

```
%S532I G71 * ......................................... Begin the program
N10 G30 G17 X+0 Y+0 Z–20 * ........................... Define the workpiece blank
N20 G31 G90 X+100 Y+100 Z+0 *
N30 G99 T25 L+0 R+15 * ............................... Define the tool
N40 T25 G17 S1500 * .................................. Call the tool
N50 G00 G40 G90 Z+100 M06 * .......................... Retract and insert tool
N60 I+50 J+50 * ...................................... Set pole
N70 G10 R+70 H+280 * ................................. Pre-position in the working plane with
                                                       polar coordinates
N80 Z–5 M03 * ........................................ Move tool to working depth
N90 G11 G41 R+50 H–90 F100 * ......................... Approach the contour with radius compensation
                                                       at machining feed rate
N100 G26 R10 * ....................................... Soft (tangential) approach
N110 G12 H+270 * ..................................... Circle to end point H = 270°, negative direction
                                                       of rotation
N120 G27 R10 * ....................................... Soft (tangential) departure
N130 G10 G40 R+70 H–110 * ............................ Depart contour, cancel radius compensation
N140 Z+100 M02 * ..................................... Retract in the infeed axis
N99999 %S532I G71 *
```

G16: Circular path with tangential connection

Moving on a circular path, the tool moves 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 Ⓔ

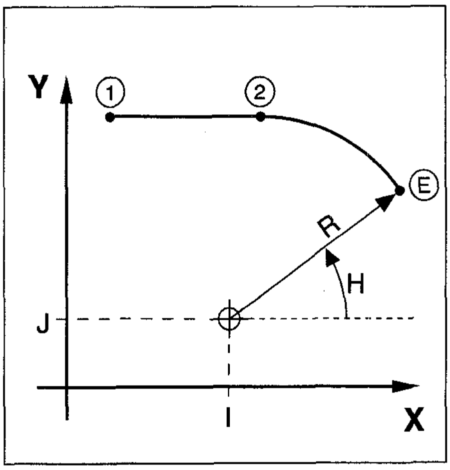


Fig. 5.37: Circular path with tangential connection around a pole

- 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 conclude the block.

If necessary, enter also:

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 a circular movement in a main plane and a linear movement perpendicular to the 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

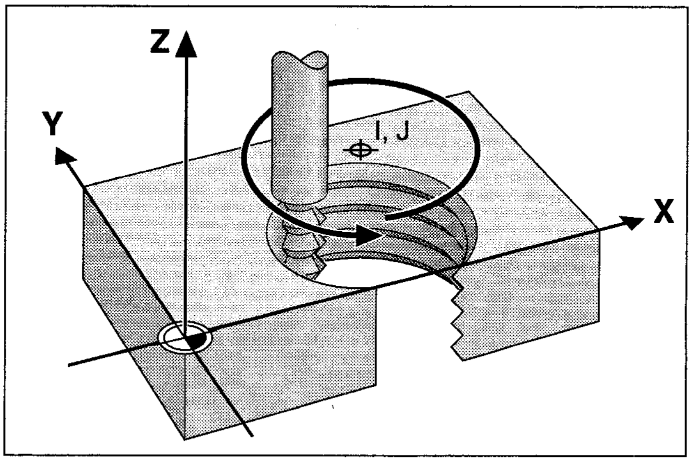


Fig. 5.38: A helix combines circular with linear motion

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 (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–	G12	G42

Fig. 5.39: The shape of the helix determines the direction of rotation and the radius compensation

To program a helix:

G 1 3	Helix, counterclockwise.
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.

If necessary, enter also:

Radius compensation
Feed rate F
Miscellaneous function M

*Resulting NC block: G13 G91 H+1080 Z+4.5 **

Example for exercise: Tapping

Given data

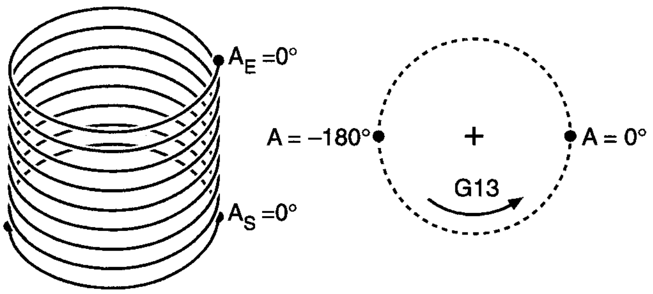
Thread:
Right-handed internal thread M64 x 1.5

Pitch P:	1.5 mm
Starting angle A_S	0°
End angle A_E:	360° = 0° at $Z_E = 0$
Thread revolutions n_R:	8

Thread overrun:

• at start of thread n_S:	0.5
• at end of thread n_E:	0.5

Number of cuts:	1



Calculating the input values

- Total height h:
  $h = P \cdot n$
  $P = 1.5$ mm
  $n = n_R + n_S + n_E = 9$
  $h = 13.5$ mm
- Incremental polar coordinate angle H:
  $H = n \cdot 360°$
  $n = 9$ (see total height h)
  $H = 360° \cdot 9 = 3240°$
- Starting angle A_S with thread overrun n_S: $n_S = 0.5$

  The starting angle of the helix is advanced by 180° (n = 1 corresponds to 360°). With positive rotation this means
  A_S with $n_S = A_S - 180° = -180°$
- Starting coordinate:
  $Z = P \cdot (n_R + n_S)$
  $= -1.5 \cdot 8.5$ 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 *
```

Part program for cutting a thread with more than 15 revolutions (also see Chapter 6)

```
   •
N80 G00 G40 G90 Z–12.75 M3
N90 G11 G41 R+32 H–180 F100
N100 G26 R+20
N110 G98 L1........................................... Identify beginning of program section repeat
N120 G13 G91 H+360 Z+1.5 F200 .......... Enter thread pitch directly as an incremental Z dimension
N130 L 1,24 ............................................ Program the number of repeats (thread revolutions)
N140 G27 R+20
   •
```

5.6 M Functions for Contouring Behavior and Coordinate Data

The following miscellaneous functions enable you to change the TNC's standard contouring behavior in certain situations:

- Smoothing corners
- Inserting rounding arcs at non-tangential straight-line transitions
- Machining small contour steps
- Machining open contours
- Programming machine-referenced coordinates

Smoothing corners: M90

Standard behavior – without M90

The TNC stops the axes briefly at sharp transitions such as inside corners and contours without radius compensation.
Advantages:

- Reduced wear on the machine
- High definition of corners (outside)

Note:

In program blocks with radius compensation (G41/G42), the TNC automatically inserts a transition arc at outside corners.

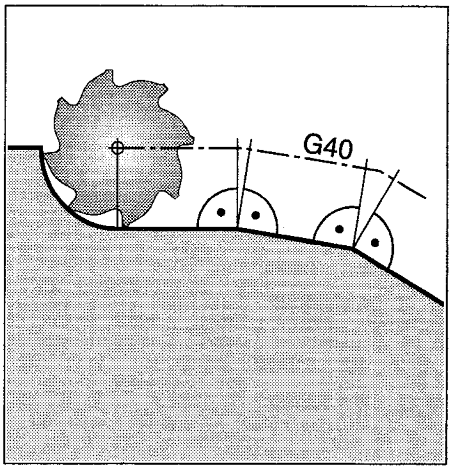


Fig. 5.40: Standard contouring behavior at G40 without M90

Smoothing corners with M90

The tool moves at corners with constant speed. Advantages:

- A smoother, more continuous surface
- Reduced machining time

Example application:
Surface consisting of a series of straight line segments.

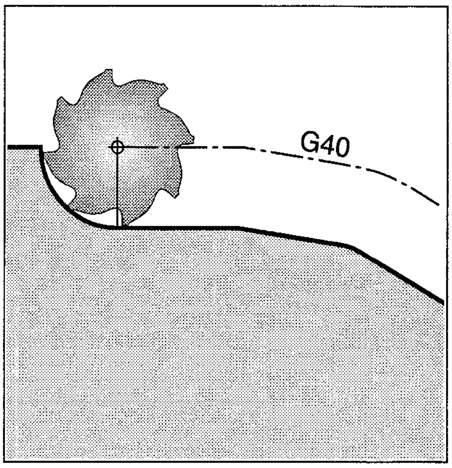


Fig. 5.41: Behavior at G40 with M90

Duration of effect

Servo lag mode must be selected. M90 is only effective in the blocks in which it is programmed.

In machine parameter MP7460 (see page 11-7) you can define an angle up to which the tool always moves at constant speed. This value is effective both with servo lag and feed precontrol.

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.

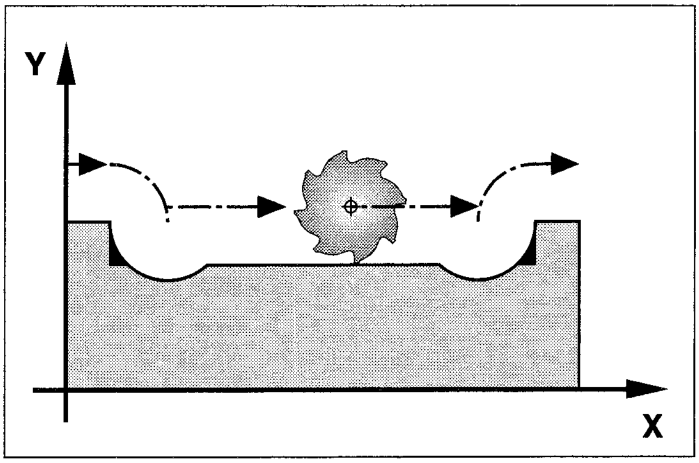


Fig. 5.42: Standard contouring behavior without M97 if the control would not generate an error message

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.

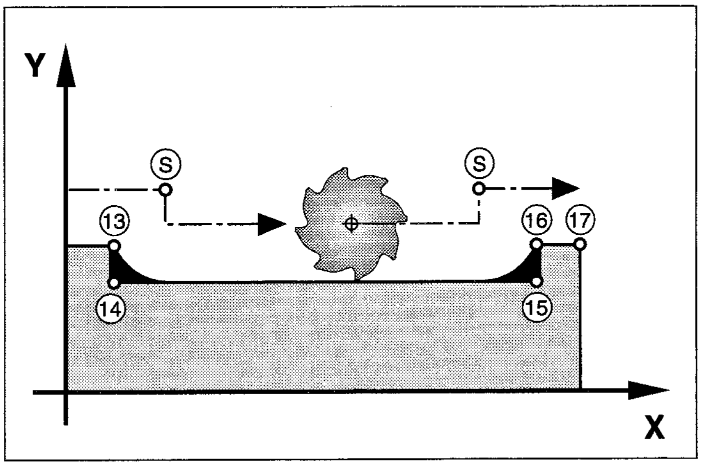


Fig. 5.43: Contouring behavior with M97

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 example

```
.
.
.
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 tool 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.

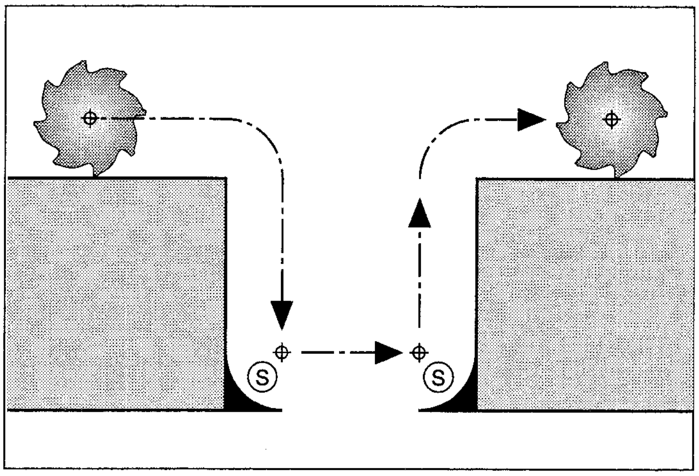


Fig. 5.44: Tool path without M98

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.

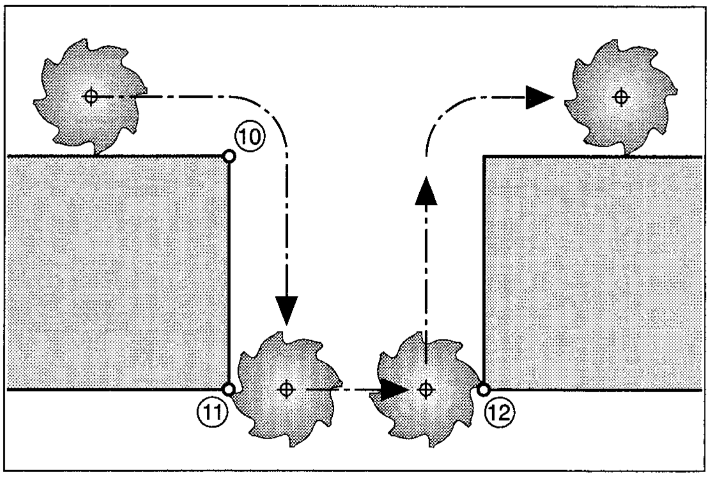


Fig. 5.45: Tool path with M98

Program example

```
.
.
.
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
.
.
.
```

Datums for coordinates: M91/M92

Standard setting

Coordinates are referenced to the workpiece datum (see page 1-17).

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 machine datum is identical with the scale reference point.

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.

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.

Workpiece datum

The user enters the coordinates of the datum for workpiece machining in the MANUAL OPERATION mode (see page 2-7).

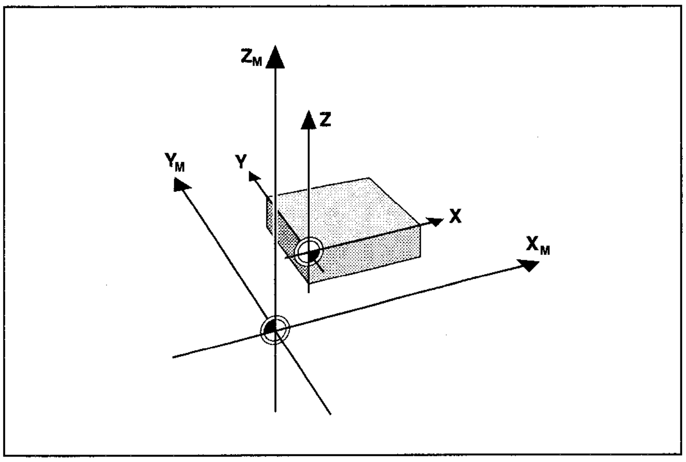


Fig. 5.46: Machine- ⊕ and workpiece datum ⊕

Reducing rotary axis display values to under 360°: M94

Machine parameter 7470 determines whether the position display value of a rotary axis are limited to

- +/– 360° or
- +/– 30 000°

If the display values are limited to +/– 30 000°, you can reduce the display value to under 360° by entering the miscellaneous function M94.

Example: M94	Only reduce display
G00 C+538 M94	Reduce display and then move to programmed value

Duration of effect

M94 is effective at the start of block and only in the block in which it is programmed.

Optimized traverse of rotary axes: M126

Standard behavior – without M126

If the display of a rotary axis is reduced to a value under 360°, the TNC will move the tool on the following path:

Actual position	Nominal position	Actual path of traverse
350°	10°	–340°
10°	340°	+330°

Optimized traverse of rotary axes – with M126

If the display of a rotary axis is reduced to a value under 360°, the TNC will move the tool on the following path:

Actual position	Nominal position	Actual path of traverse
350°	10°	+20°
10°	340°	–30°

Resulting NC block: L C+10 A+340 R0 F500 M126

Duration of effect

M126 is effective at the start of block. M126 is cancelled by M127 or at the end of the program.

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.

5.7 Positioning with Manual Data Input (MDI)

In the POSITIONING WITH MANUAL DATA INPUT mode you can enter and execute positioning blocks with G00, G01 or G07. The entered positioning blocks are not stored.

Application examples

- Pre-positioning
- Face milling
- Moving to the tool-change position

POSITIONING WITH MANUAL DATA INPUT

Select the POSITIONING WITH MDI mode of operation.

G 7	Paraxial positioning block.
G 9 0	Programming in absolute dimensions.
X 5	Position in X to 5 mm.
F 1 5 0 0 END	Set feed rate F=1500 mm/min, conclude block.
	Start the positioning block.

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

Principle

The main program is executed up to the block in which a subprogram is called with Ln,0 (①).

The subprogram is then executed from beginning to end (G98 L0) (②).

The main program is then resumed from the block after the subprogram call (③).

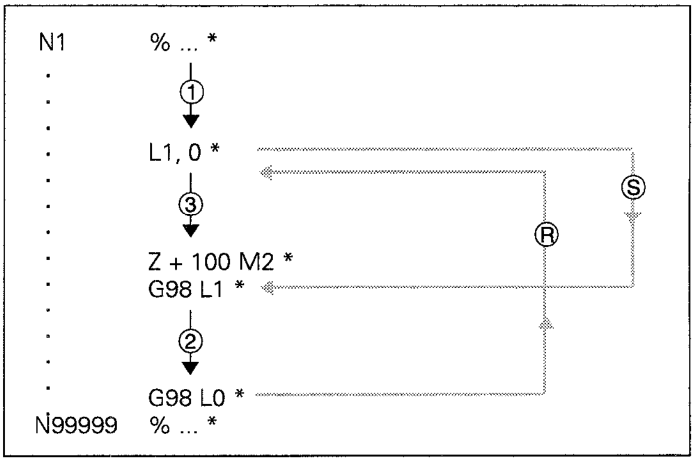


Fig. 6.1: Flow diagram for subprogramming
Ⓢ= jump Ⓡ= return jump

Operating limitations

- A main program can contain up to 254 subprograms.
- Subprograms can be called in any sequence and as often as desired.
- A subprogram cannot call itself.
- Subprograms should be written at the end of the main program (behind the block with M02 or M30).
- If subprograms are located before the block with M02 or M30, they will be executed at least once even if they are not called.

Programming and calling subprograms

Mark the beginning:

G 9 8 ENT	Select the label setting function.

LABEL NUMBER?	
5 END	The subprogram begins with (for example) label number 5.

Resulting NC block: G98 L5 *

Mark the end:

A subprogram always ends with label number 0.

G 9 8 ENT	Select the label setting function.

LABEL NUMBER?	
0 END	End of subprogram.

Resulting NC block: G98 L0 *

Call the subprogram:

A subprogram is called by its label number.

L 5 • 0 END	Call the subprogram behind label 5.

Resulting NC block: L5,0 *

The command L0,0 is not permitted (label 0 is only used to mark the end of a subprogram).

Example for exercise: Group of four holes at three different locations

The holes are drilled with Cycle G83 PECKING. Enter the setup clearance, feed rate, etc. in the cycle once. You can then call the cycle with miscellaneous function M99 (see page 8-3).

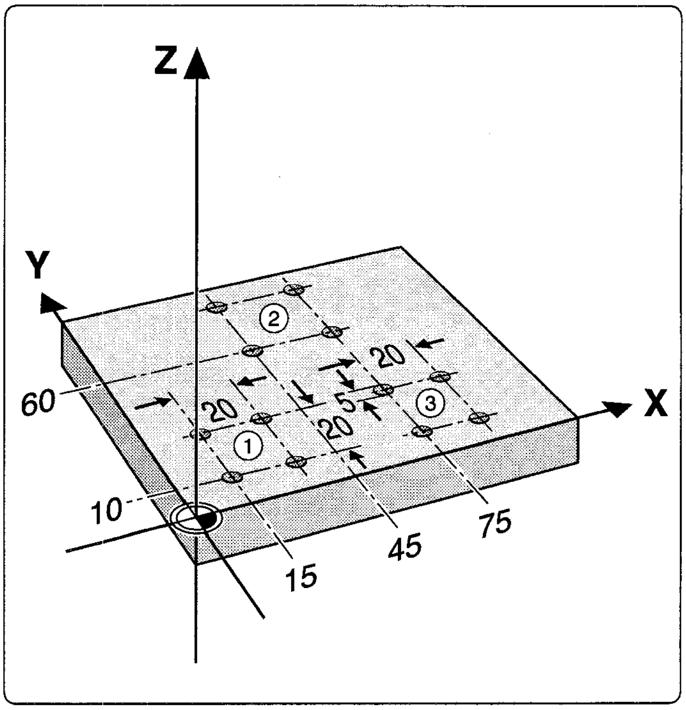
Coordinates of the first hole in each group:

Group ① X = 15 mm Y = 10 mm
Group ② X = 45 mm Y = 60 mm
Group ③ X = 75 mm Y = 10 mm

Hole spacing: X = 20 mm
Y = 20 mm

Total hole depth: Z = 10 mm

Hole diameter: Ø = 5 mm



Part program

```
%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.

Principle

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).

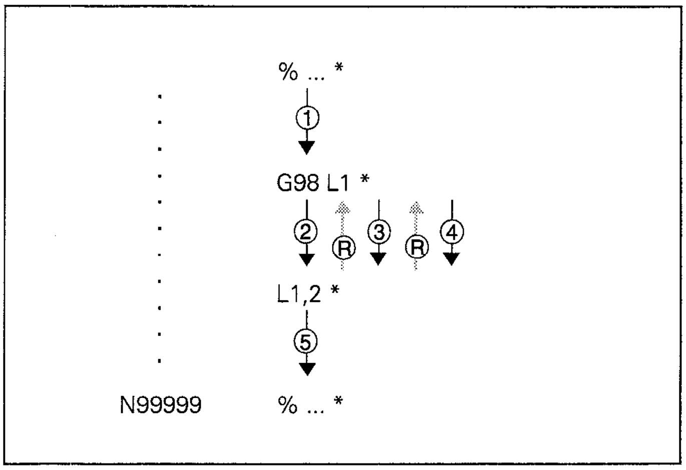


Fig. 6.2: Flow diagram for a program section repeat; Ⓡ = return jump

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 calling 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 11 times.

*Resulting NC block: L7,10 **

Example for exercise: Row of holes parallel to the X axis

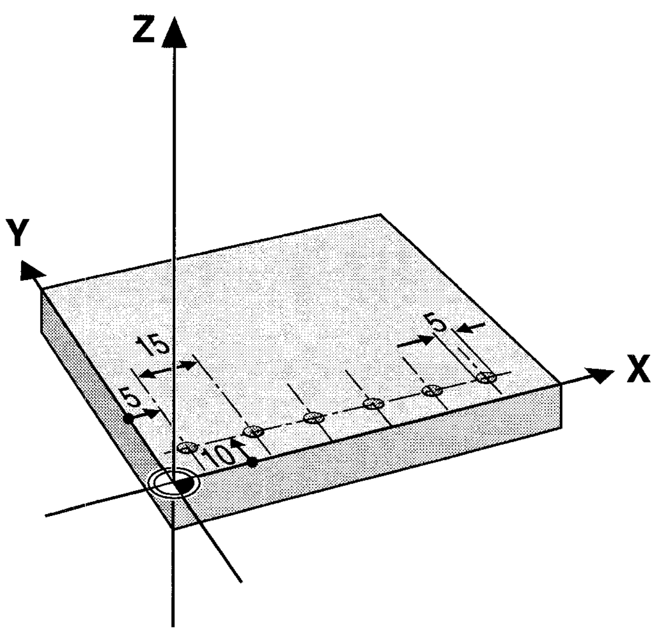
Coordinates of the first hole: X = 5 mm
Y = 10 mm

Hole spacing: IX = 15 mm

Number of holes: N = 6

Depth: Z = 10

Hole diameter: Ø = 5 mm



Part program

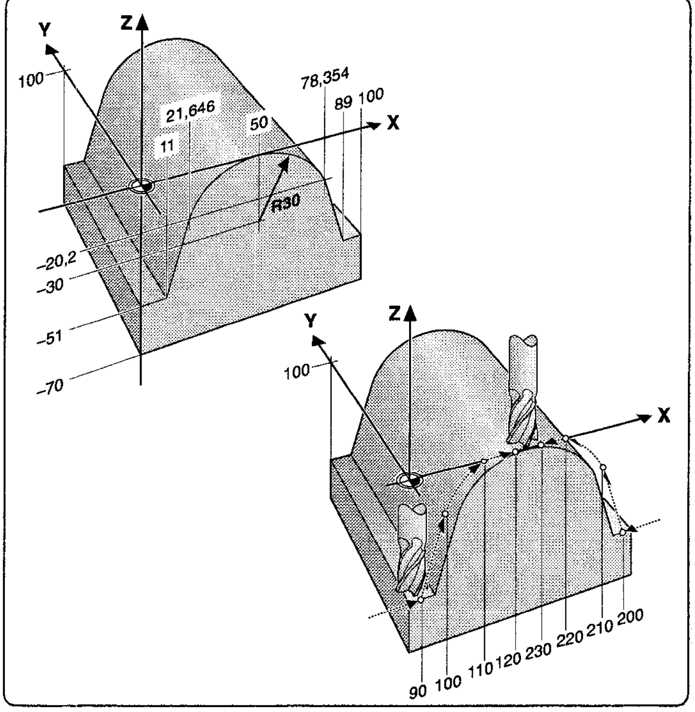
```
%S66I G71 * ............................................... Start program
N10 G30 G17 X+0 Y+0 Z-20 * ................................ Define blank form
N20 G31 G90 X+100 Y+100 Z+0 *
N30 G99 T1 L+0 R+2.5 * .................................... Define tool
N40 T1 G17 S3500 * ........................................ Call tool
N50 G00 G40 G90 Z+100 M06 * ............................... Retract and insert tool
N60 X-10 Y+10 Z+2 M03 * ................................... Pre-position to the point which is offset in negative X
                                                            direction by the hole spacing
N70 G98 L1 * .............................................. Start of the program section to be repeated
N80 G91 X+15 * ............................................ Move to drilling position (incremental dimension)
N90 G01 G90 Z-10 F100 * ................................... Drill (absolute dimension)
N100 G00 Z+2 * ............................................ Retract
N110 L1,5 * ............................................... Call LABEL 1; repeat program section from block N70 to
                                                            block N110 five times (total of six holes)
N120 Z+100 M02 * .......................................... Retract in the infeed axis
N99999 %S66I G71 *
```

Example for exercise: Milling without radius compensation using program section repeats

Sequence:

- Upward milling direction
- Machine the area from X=0 to 50 mm (program all X coordinates with the tool radius *subtracted*) and from Y=0 to 100 mm: G98 L1
- Machine the area from X=50 to X=100 mm (program all X coordinates with the tool radius *added*) and from Y=0 to 100 mm: G98 L2
- After each upward pass, the tool is moved by an increment of +2.5 mm in the Y axis.

The illustration at right shows the block numbers containing the end points of the corresponding contour elements.



Part program

```
%S67I G71 * .................................................. Start program
N10 G30 G17 X+0 Y+0 Z-70 * ................................... Define blank form (note new values)
N20 G31 G90 X+100 Y+100 Z+0 *
N30 G99 T1 L+0 R+10 * ........................................ Define tool
N40 T1 G17 S1750 * ........................................... Call tool
N50 G00 G40 G90 Z+100 M06 * .................................. Retract and insert tool
N60 X-20 Y-1 M03 * ........................................... Pre-position in the plane
N70 G98 L1 * ................................................. Start of program section 1
N80 G90 Z-51 *
N90 G01 X+1 F100 *
N100 X+11.646 Z-20.2 * ....................................... Program section for machining from
N110 G06 X+40 Z+0 * .......................................... X = 0 to 50 mm and Y = 0 to 100 mm
N120 G01 X+41 *
N130 G00 Z+10 *
N140 X-20 G91 Y+2.5 *
N150 L1,40 * ................................................. Call LABEL 1, repeat program section from block
                                                               N70 to N150 forty times
N160 G90 Z+20 * .............................................. Retract in the infeed axis
N170 X+120 Y-1 * ............................................. Pre-position for program section 2
N180 G98 L2 * ................................................ Start of program section 2
N190 G90 Z-51 *
N200 G01 X+99 F100 *
N210 X+88.354 Z-20.2 * ....................................... Program section for machining from
N220 G06 X+60 Z+0 * .......................................... X = 50 to 100 mm and Y = 0 to 100 mm
N230 G01 X+59 *
N240 G00 Z+10 *
N250 X+120 G91 Y+2.5 *
N260 L2,40 * ................................................. Call LABEL 2, repeat program section from block
                                                               N180 to N260 forty times
N270 G90 Z+100 M02 * ......................................... Retract in the infeed axis
N99999 %S67I G71 *
```

6.3 Program as Subprogram

Principle

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).

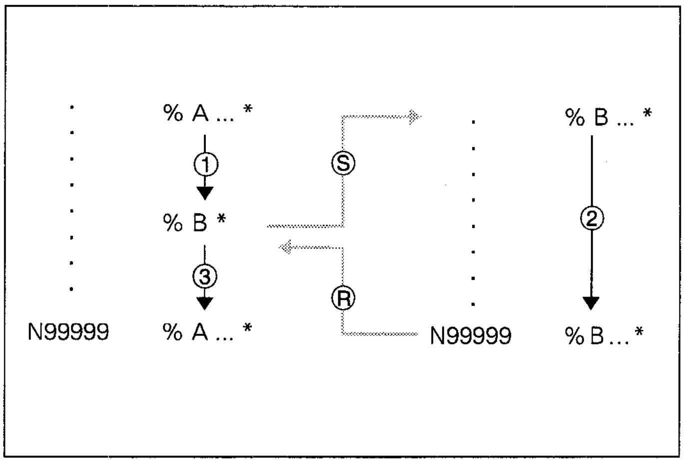


Fig. 6.3: Flow diagram of a program as subprogram
Ⓢ = jump, Ⓡ = return jump

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 program as a subprogram

% ▶ **PROGRAM NAME ?**

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

Function	Soft key
Call an externally stored program	EXT
Call a plain-language program (default setting)	.H
Call an ISO program	.I
Call a program from the TNC memory (default setting)	INT

Resulting NC block: % NAME

- You can also call a main program with cycle G39 (see page 8-38).
- When calling an ISO program, the program name must not contain G50, G70 or G71.

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

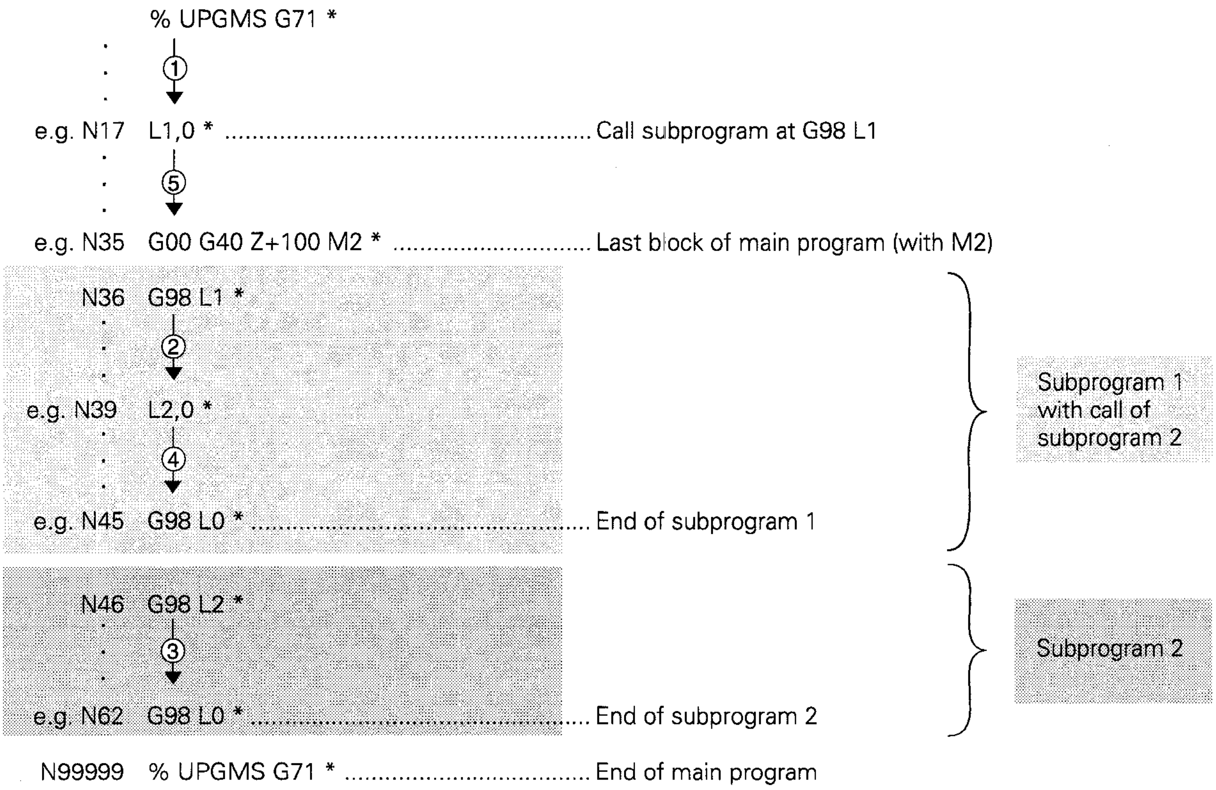
The nesting depth is the number of successive levels in which program sections or subprograms can call further program sections or subprograms.

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
      .
e.g. N39   L2,0 *
      .
      .        ④
      .
e.g. N45   G98 L0 * ....................................... End of subprogram 1
     N46   G98 L2 *
      .
      .        ③
      .
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

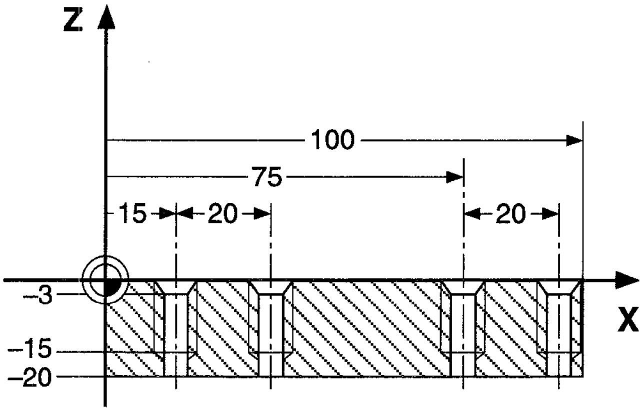
② X = 45 mm Y = 60 mm

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

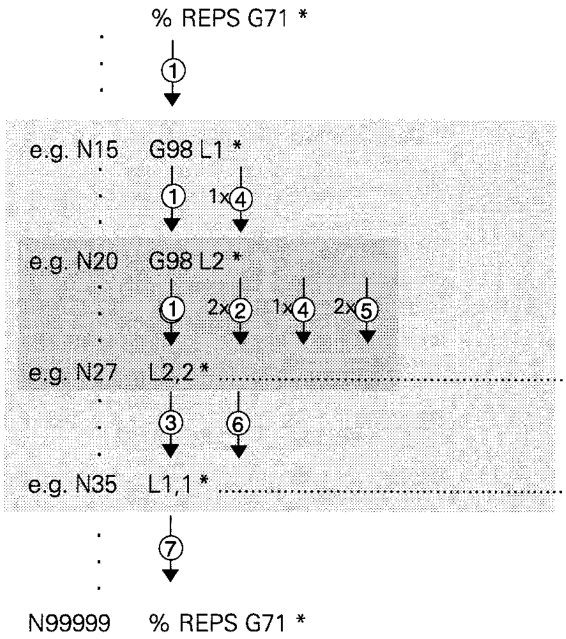


Part program

```
%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



L2,2 * .... The program section between this block and G98 L2 (block 20) is repeated twice

L1,1 * .... The program section between this block and G98 L1 (block 15) is repeated once

Sequence of 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.

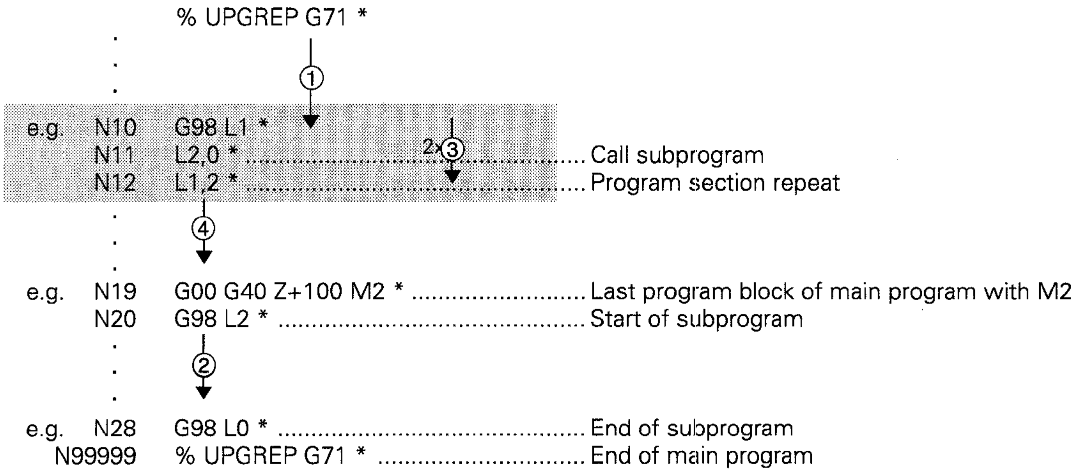
5th step: Repetition of the second step within step ④.

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 layout



```
              % UPGREP G71 *
         .
         .
         .
e.g.  N10    G98 L1 *
      N11    L2,0 * ................................... Call subprogram
      N12    L1,2 * ................................... Program section repeat
         .
         .
         .
e.g.  N19    G00 G40 Z+100 M2 * ....................... Last program block of main program with M2
      N20    G98 L2 * ................................. Start of subprogram
         .
         .
         .
e.g.  N28    G98 L0 * ................................. End of subprogram
    N99999   % UPGREP G71 * ........................... End of main program
```

Sequence of program execution

1st step: Main program UPGREP is executed up to block 11.

2nd step: Subprogram 2 is called and executed.

3rd step: Program section from block 12 to block 10 is repeated twice, so subprogram 2 is repeated twice.

4th step: Main program UPGREP is executed from block 13 to block 19. End of program.

7 Programming with Q Parameters

Q Parameters are used for:

- **Programming families of parts**
- **Defining contours through mathematical functions**

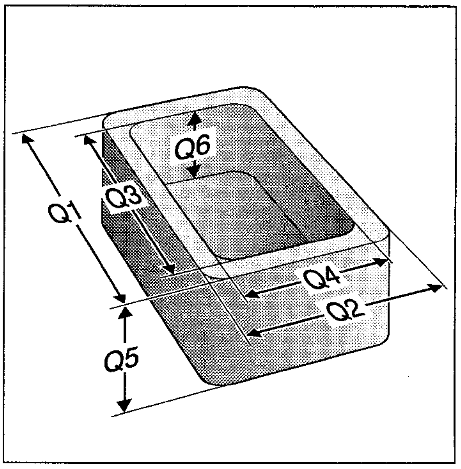


Fig. 7.1: Q parameters as variables

A **family of parts** can be programmed in the TNC 370 with a **single part program**. You do this by entering variables — called Q parameters — instead of numerical values.

A Q parameter is designated by the letter Q and a number between 0 and 299.

Meaning	**Range**
Freely available parameters, locally effective (depending on MP7251)	Q0 to Q99
Parameters for special functions on the TNC	Q100 to Q199
Parameters that are primarily reserved for cycles, globally effective	Q200 to Q299

Q parameters can represent for example:

- Coordinate values
- Feed rates
- Spindle speeds
- Cycle data

Q parameters also enable you to program **contours** that are defined through **mathematical functions**.

With Q parameters you can make the execution of machining steps dependent on **logical conditions**.

Q parameters and numerical values can also be **mixed** within a program.

Q parameters can be assigned numerical values between –99999.9999 and +99999.9999.

The TNC 370 automatically assigns data to some Q parameters. For example, parameter Q108 is assigned the current tool radius. You will find a list of these parameters in Chapter 11.

7.1 Part Families — Q Parameters in Place of Numerical Values

The Q parameter function D0: ASSIGN is used for assigning numerical values to Q parameters.
Example: N10 D00 Q10 P01+25 *

This enables you to enter variable Q parameters in the program instead of numerical values.
Example: G00 G40 G90 X + Q10 (corresponds to X + 25)

For part families, the characteristic workpiece dimensions can be programmed as Q parameters. Each of these parameters is then assigned a different value when the parts are machined.

Example

Cylinder with Q parameters

Cylinder radius R = Q1
Cylinder height H = Q2

Cylinder Z1: Q1 = +30
Q2 = +10

Cylinder Z2: Q1 = +10
Q2 = +50

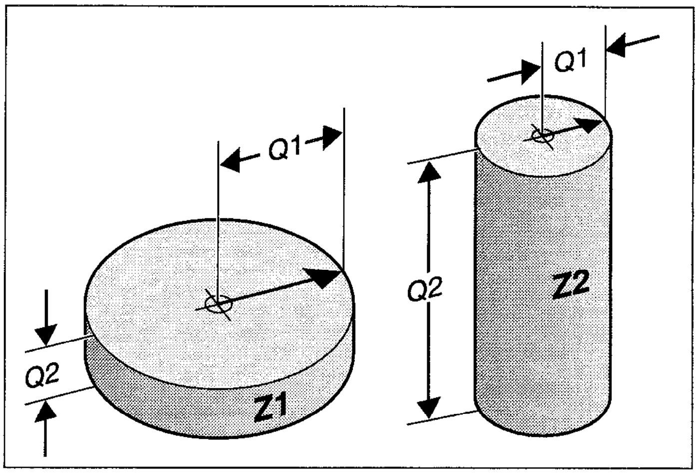


Fig. 7.2: Workpiece dimensions as Q parameters

To assign numerical values to Q parameters:

D 0 ENT	Select function D0: ASSIGN.

PARAMETER NUMBER FOR RESULT?

e.g. 5 ENT	Enter Q parameter number.

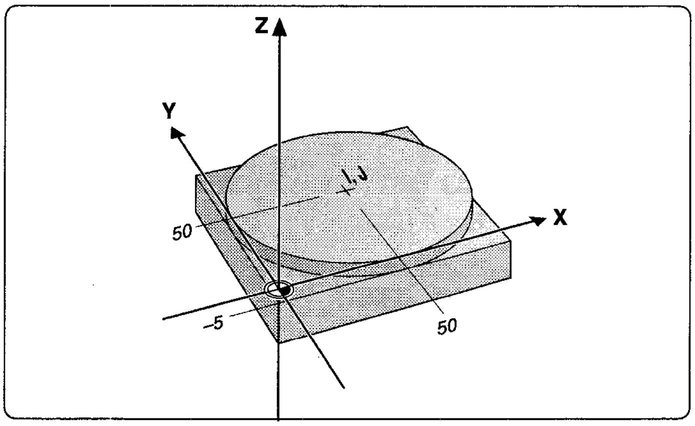
FIRST VALUE / PARAMETER?

e.g. 6 END ☐	Enter value or another Q parameter whose value is to be assigned to Q5.

*Resulting NC block: N20 D00 Q05 P01 +6 **

Example for exercise: Full circle

Circle center I,J:
X = 50 mm Y = 50 mm
Beginning and end of the circular arc:
X = 50 mm Y = 0 mm
Milling depth: ZM = –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 * ......................... Definition of blank form MIN point
N20 G30 G90 X+100 Y+100 Z+0 * ...................... Definition of blank form MAX point
N30 G99 T6 L+0 R+15 * .............................. Tool definition
N40 T6 G17 S500 * .................................. Tool call
N50 I+50J+50 * ..................................... Coordinates of the circle center
N60 G00 G40 G90 Z+100 M06 * ........................ Retract the spindle and insert the tool
N70 X+30 Y–20 * .................................... Pre-position the tool
N80 Z–5 M03 * ...................................... Pre-position the tool to working depth
N90 G01 G41 X+50 Y+0 F100 * ........................ Move to first contour point with radius compensation
N100 G02 X+50 Y+0 * ................................ Mill circular arc around circle center I,J; coordinates of end
                                                     point X = +50 and Y = 0; positive direction of rotation (G02)
N110 G00 G40 X+70 Y–20 * ........................... Retract the tool in X, Y; cancel radius compensation
N120 Z+100 M02 * ................................... Retract the tool in Z
N9999 %S520I G71 *
```

Part program with Q parameters

```
%3600741 G71 *
N10 D00 Q01 P01 +100 * ............................. Clearance height
N20 D00 Q02 P01 +30 * .............................. Start pos. X
N30 D00 Q03 P01 –20 * .............................. Start-End pos. Y
N40 D00 Q04 P01 +70 * .............................. End pos. X
N50 D00 Q05 P01 –5 * ............................... Milling depth
N60 D00 Q06 P01+50 * ............................... Circle center X
N70 D00 Q07 P01 +50 * .............................. Circle center Y
N80 D00 Q08 P01 +50 * .............................. Circle start point X
N90 D00 Q09 P01 +0 * ............................... Circle start 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
```

Blocks N10 to N120:
Assign numerical values to the Q parameters

```
N130 G30 G17 X+0 Y+0 Z–20 *
N140 G31 G90 X+100 Y+100 Z+0 *
N150 G99 T1 L+Q10 R+Q11 *
N160 T1 G17 S500 *
N170 I+Q6 J+Q7 *
N180 G00 G40 G90 Z+Q1 M06 *
N190 X+Q2 Y+Q3 *
N200 Z+Q5 M03 *
N210 G01 G41 X+Q8 Y+Q9 FQ20 *
N220 G02 X+Q8 Y+Q9 *
N230 G01 G40 X+Q4 Y+Q3 *
N240 Z+Q1 M02 *
N9999 %3600741 G71 *
```

Blocks N130 to N240:
Corresponding to blocks N10 to N120 from program S520I

7.2 Describing Contours Through Mathematical Functions

Overview

Mathematical functions assign the results of one of the following operations to a Q parameter:

Function
D00: ASSIGN Example: N10 D00 Q05 P01 +60 * Assigns a value directly
D01: ADDITION Example N10 D01 Q01 P01 –Q2 P02 –5 * Calculates and assigns the sum of two values
D02: SUBTRACTION Example: N10 D02 Q01 P01 +10 P02 +5 * Calculates and assigns the difference between two values
D03: MULTIPLICATION Example: N10 D03 Q02 P01 +3 P02 +3 * Calculates and assigns the product of two values
D04: DIVISION Example: N10 D04 Q04 P01 +8 P02 +Q02 * Calculates and assigns the quotient of two values **Not permitted: division by 0**
D05: SQUARE ROOT Example: N10 D05 Q20 P01 +4 * Calculates and assigns the square root of a number **Not permitted: square root of a negative number**

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 fundamental operations

Assign the value 10 to parameter Q5, and assign the product of Q5 and 7 to parameter Q12.

D 0 ENT	Select Q parameter function D00 (ASSIGN).
PARAMETER NUMBER FOR RESULT ?	
5 ENT	Enter parameter number, for example 5, and confirm.
FIRST VALUE / PARAMETER ?	
1 0 END □	Assign numerical value to Q5, terminate block.

D 0 ENT	Select Q parameter function D03 (MULTIPLICATION).
PARAMETER NUMBER FOR RESULT ?	
1 2 ENT	Enter parameter number, for example Q12, and confirm.
FIRST VALUE OR PARAMETER ?	
Q 5 ENT	Enter Q5 (=10) and confirm.
MULTIPLIER ?	
7 END □	Enter 7, terminate block.

*Resulting NC blocks: N20 D00 Q05 P01 +10 **
*N30 D03 Q12 P01 +Q5 P02 +7 **

7.3 Trigonometric Functions

Sine, *cosine* and *tangent* are terms designating the ratios of the sides of right triangles.

For a right triangle, the trigonometric functions of the angle α are defined by the equations

$$\sin\alpha = a/c,$$
$$\cos\alpha = b/c,$$
$$\tan\alpha = a/b = \sin\alpha / \cos\alpha,$$

where

- c is the side opposite the right angle
- a is the side opposite angle α
- b the third side.

The angle can be found from the tangent:

$$\alpha = \text{arc tan } \alpha = \text{arc tan } (a/b) = \text{arc tan } (\sin\alpha / \cos\alpha)$$

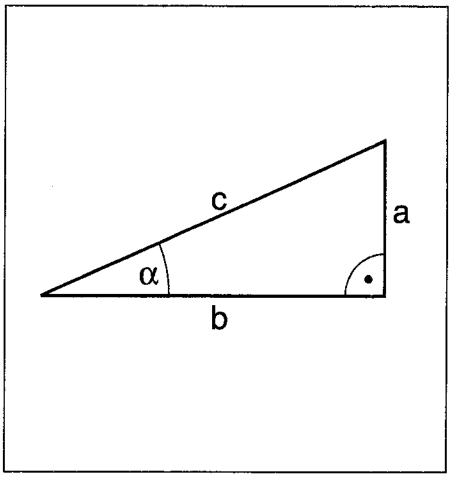


Fig. 7.3: Sides and angles on a right triangle

Example: a = 10 mm
b = 10 mm
α = arctan (a/b) = arctan 1 = 45°

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

Overview

Function
D06: SINE Example: N10 D06 Q20 P01 –Q05 * Calculate the sine of an angle in degrees (°) and assign it to a parameter
D07: COSINE Example: N10 D07 Q21 P01 –Q05 * Calculate the cosine of an angle in degrees (°) and assign it to a parameter
D08: ROOT SUM OF SQUARES Example: N10 D08 Q10 P01 +5 P02 +4 * Take the square root of the sum of two squares and assign it to a parameter
D13: ANGLE Example: N10 D13 Q20 P01 +10 P02 –Q01 * Calculate the angle from the arc tangent of two sides or from the sine and cosine of the angle, and assign it to a parameter

7.4 If-Then Operations with Q Parameters

If-Then conditional operations enable the TNC to compare a Q parameter with another Q parameter or with a numerical value.

Jumps

The jump target is specified in the block through a label number. If the programmed condition is true, the TNC continues the program at the specified label; if it is false, the next block is executed.

To jump to another program, you enter a program call after the block with the target label (see page 6-8).

Overview

Function
D09: IF EQUAL, JUMP Example: N10 D09 P01 +Q01 P02 +Q03 P03 5 * If the two values or parameters are equal, go to the specified label (here, label 5)
D10: IF NOT EQUAL, JUMP Example: N10 D10 P01 +10 P02 –Q05 P03 10 * If the two values or parameters are not equal, go to the specified label (here, label 19)
D11: IF GREATER THAN, JUMP Example: N10 D11 P01 +Q01 P02 –10 P03 5 * If the first value or parameter is larger than the second, go to the specified label (here, label 5)
D12: IF LESS THAN, JUMP Example: N10 D12 P01 +Q05 P02 +0 P03 1 * If the first value or parameter is less than the second, go to the specified label (here, label 1)

Unconditional jumps

Unconditional jumps are jumps which are always executed because the condition is always true.
Example:

N20 D09 P01 +10 P02 +10 P03 1 *

Program example

When Q5 becomes negative, a jump to program 100 will occur.

```
.
.
.
N50 D00 Q05 P01 +10 * ........................................... Assign value, for example 10, to parameter Q5
.
.
.
N90 D02 Q05 P01 +Q5 P02 +12 * ............................ Reduce the value of Q5
N100 D12 P01 +Q5 P02 +0 P03 5 * .......................... If +Q5 is less than 0, jump to label 5
.
.
.
N150 G98 L5 * .......................................................... Label 5
N160 %100 * ............................................................ Jump to program 100
.
.
.
```

7.5 Checking and Changing Q Parameters

During a program run or test run, Q parameters can be checked and changed if necessary.

Preparation:

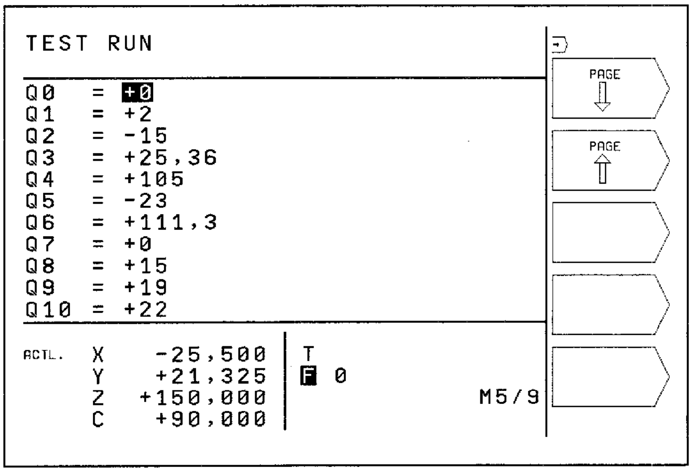
- ➤ A running program must be aborted (such as by pressing the machine STOP button and STOP key)
- ➤ If you are doing a test run, interrupt it

To call a Q parameter

You can call a Q parameter table by pressing the black Q-key (see figure) in which the currently active Q parameter values are filed.

Use the black arrow keys to select a Q parameter on a screen page. With the arrow soft keys you can move to the next or previous page of the table.

You can change a Q parameter value by overwriting the old value. Press END to exit the table and save all changed values.



```
TEST RUN

Q0  = +0
Q1  = +2
Q2  = -15
Q3  = +25,36
Q4  = +105
Q5  = -23
Q6  = +111,3
Q7  = +0
Q8  = +15
Q9  = +19
Q10 = +22

ACTL.  X    -25,500 | T
       Y    +21,325 | F 0
       Z   +150,000 |                 M5/9
       C    +90,000 |
```

Fig. 7.4: Q parameter table

7.6 Output of Q Parameters and Messages

Displaying error messages

With the function D14: ERROR NUMBER 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 interrupts the run and displays an error message. The program must then be restarted.

Input example:

N50 D14 P01 254 *

The TNC will display the text of error number 254.

Error number to be entered	Prepared dialog text
0 to 299	FN 14: ERROR CODE 0 to 299
300 to 399	PLC: ERROR 0 to 99
400 to 499	DIALOG 0 to 99
1000 to 1099	Internal error messages (see table below)

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

Error code	Error text
1000	SPINDLE MUST BE TURNING
1001	TOOL AXIS IS MISSING
1002	SLOT WIDTH TOOL LARGE
1003	TOOL RADIUS TOO LARGE
1004	RANGE EXCEEDED
1005	START POSITION INCORRECT
1006	ROTATION NOT PERMITTED
1007	SCALING FACTOR NOT PERMITTED
1008	MIRRORING NOT PERMITTED
1009	DATUM SHIFT NOT PERMITTED
1010	FEED RATE IS MISSING
1011	ENTRY VALUE INCORRECT

Error code	Error text
1012	WRONG SIGN PROGRAMMED
1013	ENTERED ANGLE NOT PERMITTED
1014	TOUCH POINT INACCESSIBLE
1015	TOO MANY POINTS
1016	CONTRADICTORY ENTRY
1017	CYCL INCOMPLETE
1018	PLANE WRONGLY DEFINED
1019	WRONG AXIS PROGRAMMED
1020	WRONG RPM
1021	RADIUS COMP. UNDEFINED
1022	ROUNDING-OFF UNDEFINED
1023	ROUNDING RADIUS TOO LARGE
1024	PROGRAM START UNDEFINED
1025	EXCESSIVE SUBPROGRAMMING
1026	ANGLE REFERENCE MISSING

Output through an external data interface

The function D15: PRINT transmits the values of Q parameters and error messages over the data interface. When you are transferring values to a PC, the TNC generates the file %FN15RUN.A in the PC memory to store the transferred values.

- D15: PRINT with numerical values up to 200
  Example: D15: PRINT 20
  Transmits the corresponding error message (see overview for D14).
- D15: PRINT with Q parameter
  Example: D15: PRINT Q20
  Transmits the value of the corresponding Q parameter.

Up to six Q parameters and numerical values can be transmitted simultaneously. The TNC separates them with slashes.
Example: D15: PRINT 1/Q1/2/Q2

Assigning values for the PLC

Function D19: PLC transmits up to two numerical values or Q parameters to the PLC.

Input increment and unit of measure: 1 µm or 0.001°

Example: N25 D19 P01 +10 P02 +Q3 *

The number 10 corresponds to 10 µm or 0.01°

7.7 Measuring with the 3D Touch Probe During Program Run

The 3D touch probe can measure positions on a workpiece during program run.

Applications:

- Measuring differences in the height of cast surfaces
- Checking tolerances during machining

Enter G55 to activate the touch probe.
The touch probe is automatically pre-positioned (with rapid traverse from MP6150) and probes the specified position (with feed rate from MP6120). The coordinate measured for the probe point is stored in a Q parameter.

The TNC interrupts the probing process if the probe is not deflected within a certain range (range selected with MP6130).

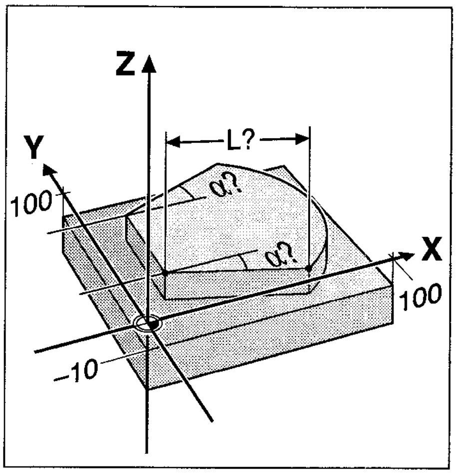


Fig. 7.4: Workpiece dimensions to be measured

To program the use of a touch probe:

G 5 5 ENT	Select the touch probe function.
PARAMETER NUMBER FOR RESULT ?	
5 ENT	Enter the number of the Q parameter to which the coordinate is to be assigned, for example Q5.
PROBING AXIS/PROBING DIRECTION?	
X – ENT	Enter the probing axis for the coordinate, for example X. Select and confirm the probing direction.
X 5 ENT Y 0 ENT Z – 5 ENT	Enter all coordinates of the pre-positioning point values, for example X = 5 mm, Y = 0, Z = –5 mm.
END □	Conclude input.

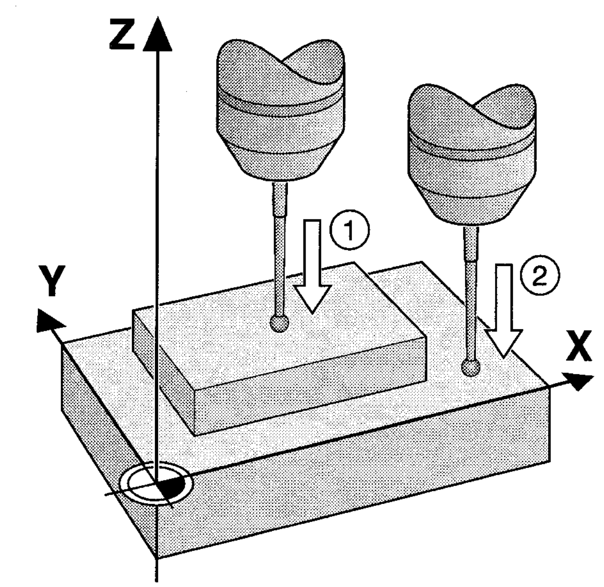
*Resulting NC block: N150 G55 P01 05 P02 X- X+5 Y+0 Z–5 **

Pre-position the touch probe manually such that it will not collide with the workpiece when it moves toward the programmed position.

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

```
%3600717 G71 *                        }
N10 D00 Q11 P01 +20 *                 }
N20 D00 Q12 P01 +50 *                 } Begin the program; assign the coordinates for pre-
N30 D00 Q13 P01 +10 *                 } positioning the touch probe to Q parameters
N40 D00 Q21 P01 +50 *                 }
N50 D00 Q22 P01 +10 *                 }
N60 D00 Q23 P01 +0 *

N70 T0 G17 *
N80 G00 G40 G90 Z+100 M06 * .............................. Insert touch probe
N90 G55 P01 10 P02 Z- X+Q11 Y+Q12 Z+Q13 * ...... The Z coordinate probed in the negative direction is stored in
                                                 Q10 (1st point)
N100 X+Q21 Y+Q22 * ............................................. Move to auxiliary point for second pre-positioning
N110 G55 P01 20 P02 Z- X+Q21 Y+Q22 Z+Q23 * .... The Z coordinate probed in the negative direction is stored in
                                                 Q20 (2nd point)
N120 D02 Q01 P01 +Q20 P02 +Q10......................... Measure the height of the island and assign to Q1
N130 G38 * ............................................................. Q1 can be checked after the program run has been stopped
                                                 (see page 7-10)
N140 Z+100 M02 *
N9999 %3600717 G71 * .......................................... Retract the tool and end the program
```

7.8 Examples for Exercise

Rectangular pocket with corner rounding and tangential approach

Pocket center coordinates

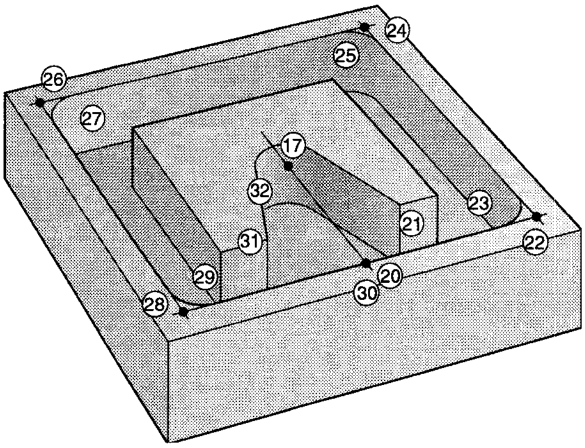
X = 50 mm (Q1)
Y = 50 mm (Q2)

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 F = 200 mm/min (Q7)

Note:

At corners 21 and 31 the workpiece will be machined slightly differently than shown in the drawing!



Part program

```
%360077 G71 *
N10 G30 G17 X+0 Y+0 Z-20 *               } Program start and workpiece blank
N20 G31 G90 X+100 Y+100 Z+0 *
N30 D00 Q01 P01 +50 *                    
N40 D00 Q02 P01 +50 *
N50 D00 Q03 P01 +90 *
N60 D00 Q04 P01 +70 *                    } Assign the rectangular pocket data to Q parameters
N70 D00 Q05 P01 +15 *
N80 D00 Q06 P01 +10 *
N90 D00 Q07 P01 +200 *
N100 G99 T1 L+0 R+5 *
N110 T1 G17 S1000 *                      } Define and insert the tool
N120 G00 G40 G90 Z+100 M6 *
N130 D04 P01 Q13 P02 +Q03 P03 +2 *       } Enter half the pocket length and width for the paths of
N140 D04 P01 Q14 P02 +Q04 P03 +2 *       } traverse in blocks N200, N220, N300
N150 D04 P01 Q16 P02 +Q06 P03 +4 * ..................Rounding radius for smooth approach
N160 D04 P01 Q17 P02 +Q07 P03 +2 * ..................Feed rate in corners is half the rate for linear movement
N170 X+Q01 Y+Q02 M03 * ..............................Pre-position in X and Y (pocket center), spindle ON
N180 Z+2 * ..........................................Pre-position over workpiece
N190 G01 Z-Q05 FQ07 * ...............................Move to working depth Q5 (= –15 mm) with feed rate Q7
                                                      (= 100)
N200 G41 G91 X+Q13 G90 Y+Q02 *           } Approach the pocket in a tangential arc
N210 G26 RQ16 *
N220 G91 Y+Q14 *
N230 G25 RQ6 FQ17 *
N240 X-Q3 *
N250 G25 RQ6 FQ17 *
N260 Y-Q4 *                              } Mill the frame of the rectangular pocket
N270 G25 RQ6 FQ17 *
N280 X+Q3 *
N290 G25 RQ6 FQ17 *
N300 Y+Q14 *
N310 G27 RQ16 *                          } Depart to pocket center in a tangential arc
N320 G00 G40 G90 X+Q1 Y+Q2 *
N330 Z+100 M02 * ....................................Retract tool
N9999 %360077 G71 *
```

Bolt hole circles

Bore pattern 1 distributed over a full circle:

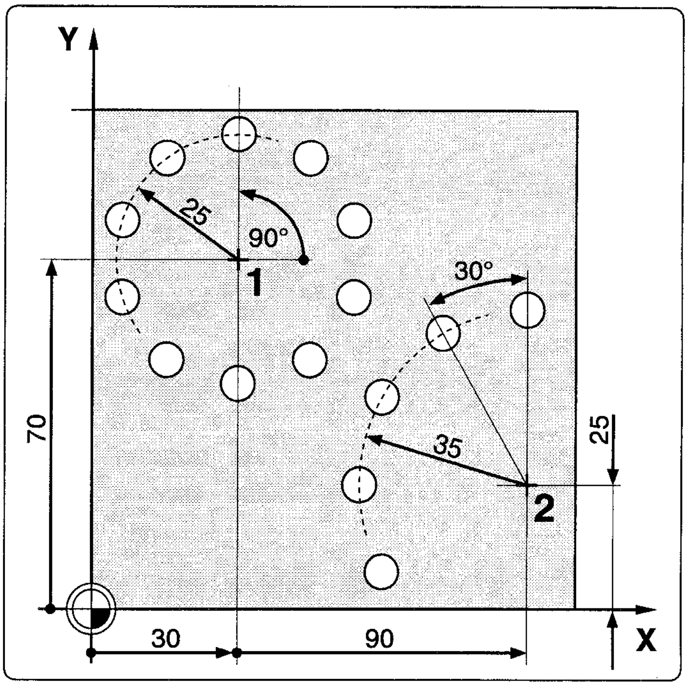
Entry values are listed below in program blocks N10 to N80.

Movements in the plane are programmed with polar coordinates.

Bore pattern 2 distributed over a circle sector:

Entry values are listed below in blocks N150 to N190; Q5, Q7 and Q8 remain the same.

The holes are executed with Cycle G83: PECKING (see page 8-4)



Part program

```
%3600715 G71 * ........................................ Load data for bolt hole circle 1
N10 D00 Q01 P01 +30 * ................................. Circle center X coordinate
N20 D00 Q02 P01 +70 * ................................. Circle center Y coordinate
N30 D00 Q03 P01 +11 * ................................. Number of holes
N40 D00 Q04 P01 +25 * ................................. Circle radius
N50 D00 Q05 P01 +90 * ................................. Starting angle
N60 D00 Q06 P01 +0 * .................................. Hole angle increment (0: distribute holes over 360°)
N70 D00 Q07 P01 +2 * .................................. Setup clearance
N80 D00 Q08 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 -Q07 ..................................... Definition of the pecking cycle/setup clearance
         P02 -Q08...................................... Total hole depth according to the load data
         P03 -5 ....................................... Pecking depth
         P04 0 ........................................ Dwell time
         P05 250 * .................................... Feed rate for 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 circle radius
N190 D00 Q6 P01 +30 * ................................. New hole angle increment (not a full circle, 5 holes at 30°
                                                        intervals)
N200 L1,0 * ........................................... Call bolt hole circle 2
N210 G00 G40 G90 Z+200 M02 * .......................... End of main program
```

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 P02 +0 P03 10 * .......................... If the hole angle increment has been entered, jump to LBL 10
N250 D04 Q6 P01 +360 P02 +Q3 * ............................ Calculate the hole angle increment, distribute holes over 360°
N260 G98 L10 *
N270 D01 Q11 P01 +Q5 P02 +Q6 * ............................ Calculate second hole position from the start angle and hole
                                                            angle increment
N280 I+Q1 J+Q2 * .......................................... Set pole at bolt hole circle center
N290 G10 G40 G90 R+Q4H+Q5 M03 * ........................... Move in the plane to 1st hole
N300 G00 Z+Q7 M99 * ....................................... Move in Z to setup clearance, call cycle
N310 D01 Q10 P01 +Q10 P02 +1 * ............................ Count finished holes
N320 D09 P01 +Q10 P02 +Q3 P03 99 * ........................ Finished?
N330 G98 L2 *
N340 G10 R+Q4 H+Q11 M99 * ................................. Make a second 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 (update)
N370 D12 P01 +Q10 P02 +Q3 P03 2 * ......................... Not finished?
N380 G98 L99 *
N390 G00 G40 G90 Z+200 * .................................. Retract in Z
N400 G98 L0 * ............................................. End of subprogram, return jump to main program
N9999 %3600715 G71 *
```

Ellipse

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

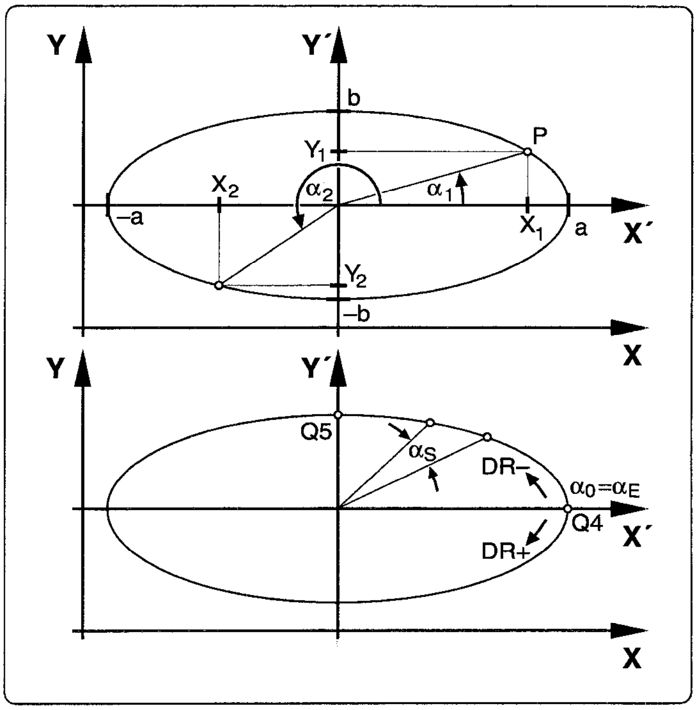
a, *b* : Semimajor and semiminor axes of the ellipse
α : Angle between the leading axis and the connecting line from *P* to the center of the ellipse

Process:

The points of the ellipse are calculated and connected by many short lines. The more points that are calculated and the shorter the lines between them, the smoother the curve.

The machining direction can be varied by changing the entries for start and end angles.

The input parameters are listed below in blocks N10 to N120 of the part program.



Part program

```
%376015 G71 * ........................................................Load data
N10 D00 Q01 P01 +50 * ............................................X coordinate for center of ellipse
N20 D00 Q02 P01 +50 * ............................................Y coordinate for center of ellipse
N30 D00 Q03 P01 +50 * ............................................Semiaxis in X
N40 D00 Q04 P01 +20 * ............................................Semiaxis in Y
N50 D00 Q05 P01 +0 * ..............................................Start angle
N60 D00 Q06 P01 +360 * ..........................................End angle
N70 D00 Q07 P01 +40 * ............................................Number of calculating steps
N80 D00 Q08 P01 +0 * ..............................................Rotational position
N90 D00 Q09 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 Z
N130 G30 G17 X+0 Y+0 Z–20 * ................................Definition of workpiece blank
N140 G31 G90 X+100 Y+100 Z+0 *
N150 G99 T1 L+0 R+2.5 *
N160 T1 G17 S2500 *
N170 G00 G40 G90 Z+100 * ....................................Retract in Z
N180 L10,0 * .............................................................Call subprogram ellipse
N190 G00 Z+100 M02 * ...........................................Retract in Z, end of main program
```

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 D02 Q35 P01 +Q6 P02 +Q5 *
N240 D04 Q35 P01 +Q35 P02 +Q7 * .......................... Calculate angle increment
N250 D00 Q36 P01 +Q5 * ................................... Current angle for calculation = set start angle
N260 D00 Q37 P01 +0 * .................................... Set counter for milled steps
N270 L11,0 * ............................................. Call subprogram for calculating the points of the ellipse
N280 G00 G40 X+Q21 Y+Q22 M03 * ........................... Move to start 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 D01 Q36 P01 +Q36 P02 +Q35 * ......................... Update the angle
N330 D01 Q37 P01 +Q37 P02 +1 * ........................... Update the counter
N340 L11,0 * ............................................. Call subprogram for calculating the points of the ellipse
N350 G01 X+Q21 Y+Q22 FQ11 * .............................. Move to next point
N360 D12 P01 +Q37 P02 +Q7 P03 1 * ........................ Not finished?

N370 G73 G90 H+0 * ....................................... Reset rotation
N380 G54 X+0 Y+0 * ....................................... Reset datum shift
N390 G00 G40 Z+Q12 * ..................................... Move in Z to setup clearance
N400 G98 L0 * ............................................ End of subprogram for milling the ellipse

N410 G98 L11 *
N420 D07 Q21 P01 +Q36 *
N430 D03 Q21 P01 +Q21 P02 +Q3 * .......................... Calculate X coordinate
N440 D06 Q22 P01 +Q36 *
N450 D03 Q22 P01 +Q22 P02 +Q4 * .......................... Calculate Y coordinate
N460 G98 L0 *
N9999 %376015 G71 *
```

Machining a hemisphere with an end mill

Notes on the program:

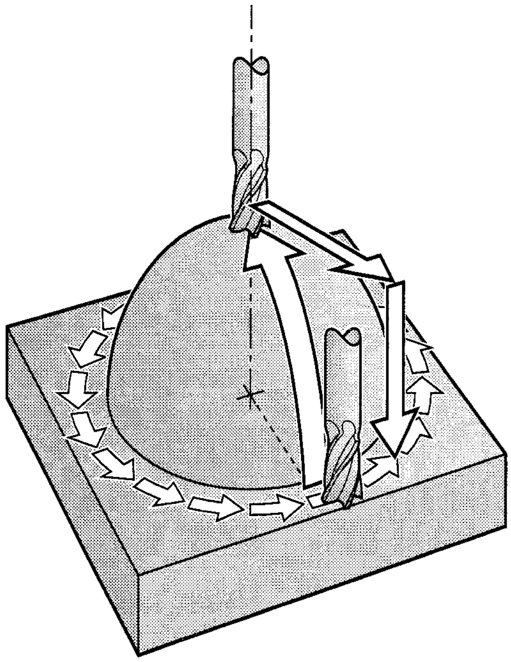
- The tool moves upwards in the ZX plane.
- You can enter an oversize in block N120 (Q12) if you want to machine the contour in several steps.
- The tool radius is automatically compensated with parameter Q108.

The program works with the following values:

• Solid angle:	Start angle	Q1
	End angle	Q2
	Increment	Q3
• Sphere radius		Q4
• Setup clearance		Q5
• Plane angle:	Start 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 center of tool
- Q26: Plane angle during machining
- Q108: TNC parameter with tool radius



Part program

```
%360712 G71 *
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 *                     Assign the sphere data to the
N70 D00 Q7 P01 + 360 *                  parameters
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 *
N130 G30 G17 X+0 Y+0 Z-50 *
N140 G31 G90 X+100 Y+100 Z+0 *
N150 G99 T1 L+0 R+5 *                   Workpiece blank; define and insert tool
N160 T1 G17 S1000 *
N170 G00 G40 G90 Z+100 M06 *
N180 L 10,0 * ......................... Subprogram call
N190 G00 G40 G90 Z+100 M02 * .......... Retract tool; return jump to beginning of program
```

Continued on next page...

```
N200 G98 L10 *
N200 D01 Q15 P01 +Q5 P02 +Q4 *          }
N220 D00 Q21 P01 +Q1 *                  }
N230 D01 Q24 P01 +Q4 P02 +Q108 *        } Determine starting and calculation values
N240 D00 Q26 P01 +Q6 *                  }
N250 G54 X+Q9 Y+Q10 Z–Q4 * .............. Shift datum to center of sphere
N260 G73 G90 H+Q06 * .................... Rotation for program start (starting plane angle)

N270 I+0 J+0 * .......................... Pole for pre-positioning
N280 G10 G40 G90 R+Q24 H+Q6 * ........... Pre-positioning before machining

N290 G98 L1 *
N300 K+0 I+Q108 *
N310 G01 Y+0 Z+0 FQ11 * ................. Pre-positioning at the beginning of each arc
N320 G98 L2 *
N330 G11 G40 R+Q4 H+Q21 FQ11 *          }
N340 D02 Q21 P01 +Q21 P02 +Q03 *        } Mill the sphere upward until the highest point is reached
N350 D11 P01 +Q21 P02 +Q02 P03 2 *      }
N360 R+Q04 H+Q02 *                      }
N370 G01 Z+Q15 F1000 *                  } Mill the highest point and then retract the tool
N380 G00 G40 X+Q24 *                    }
N390 D01 Q26 P01 +Q26 P02 +Q08 * ........ Prepare the next rotation increment
N400 D00 Q21 P01 Q01 * .................. Reset solid angle for machining to the starting value
N410 G73 G90 H+Q26 *                    } Rotate the coordinate system about the Z axis until plane end
N420 D12 P01 +Q26 P02 +Q07 P03 1 *      } angle is reached
N430 D09 P01 +Q26 P02 + Q07 P03 1 *     }
N440 G73 G90 H+0 * ...................... Reset rotation
N450 G54 X+0 Y+0 Z+0 * .................. Reset datum shift
N460 G98 L0 * ........................... End of subprogram
N9999 %360712 G71 *
```

8 Cycles

8.1 General Overview of Cycles

Standard cycles are frequently recurring machining sequences comprising several working steps that are stored in the control memory. 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, tapping, and the milling operations slot milling, rectangular pocket milling and circular pocket milling.
- **SL (Subcontour List) Cycles**. These allow machining of relatively complex contours composed of overlapping subcontours.
- **Coordinate transformation cycles**. These make it possible to rotate, mirror, enlarge and reduce contours, and to shift their datums.
- **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, for example Rigid Tapping.
SET-UP CLEARANCE ?	
e.g. 2 ENT	Enter the setup clearance (here, 2 mm).
TOTAL HOLE DEPTH ?	
e.g. – 3 0 ENT	Enter the total hole depth (here, –30 mm).
THREAD PITCH ?	
e.g. 0 . 7 5 END □	Enter the thread pitch (here, +0.75 mm).

*Resulting NC block: G85 P01 2 P02 –30 P03 +0.75 **

Dimensions in the tool axis

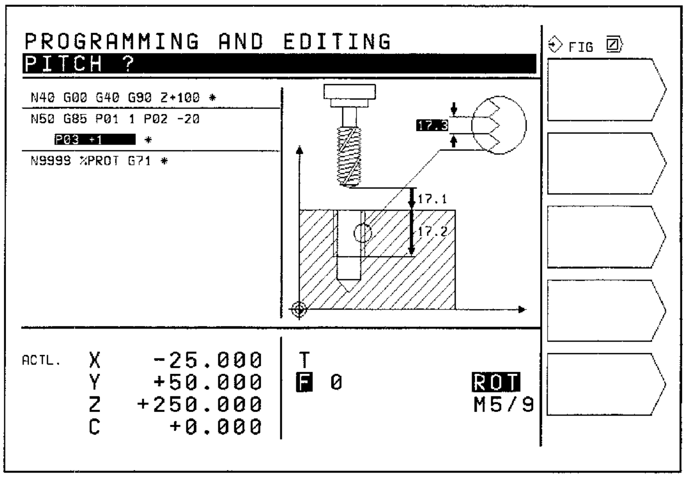
The dimensions for tool axis movement are always referenced to the position of the tool at the time of the cycle call and interpreted by the control as incremental dimensions. It is not necessary to program G91.

The algebraic sign for TOTAL HOLE DEPTH defines the working direction.

The TNC assumes that at the beginning of the cycle the tool is positioned over the workpiece at the clearance height.

Graphically assisted cycle definition

The PROGRAMMING AND EDITING mode includes graphics showing all input parameters necessary for programming a cycle.



PROGRAMMING AND EDITING
PITCH ?

```
N40 G00 G40 G90 Z+100 *
N50 G85 P01 1 P02 -20
    P03 +1     *
N9999 %PROT G71 *
```

```
ACTL.  X    -25.000   T
       Y    +50.000   F 0          ROT
       Z   +250.000                M5/9
       C     +0.000
```

Fig. 8.1: TNC graphics show the input parameters for cycle programming

To use the graphics, switch the screen to TEXT + FIGURE (see page 1-4).

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

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.

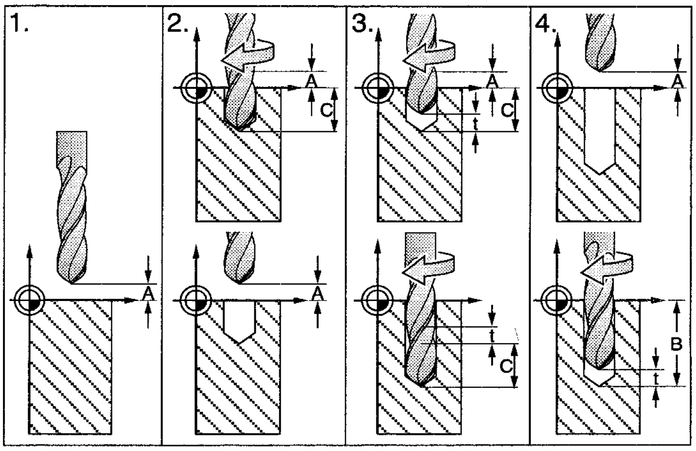


Fig. 8.2: PECKING cycle

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).
- PECKING DEPTH Ⓒ:
  Infeed per cut.
  If the TOTAL HOLE DEPTH equals the PECKING DEPTH, the tool will drill to the programmed total hole depth in one operation.
  The PECKING DEPTH does not have to be a multiple of the TOTAL HOLE DEPTH.
  If the PECKING DEPTH is programmed greater than the TOTAL HOLE DEPTH, the tool only advances to the specified TOTAL HOLE DEPTH.
- DWELL TIME in seconds:
  Amount of time the tool remains at the total hole depth for chip breaking.
- FEED RATE:
  Traversing speed of the tool during drilling.

Calculations

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

- At a total hole depth of up to 30 mm, t = 0.6 mm
- At a total hole depth exceeding 30 mm, t = total hole depth / 50
  Maximum advanced stop distance: 7 mm

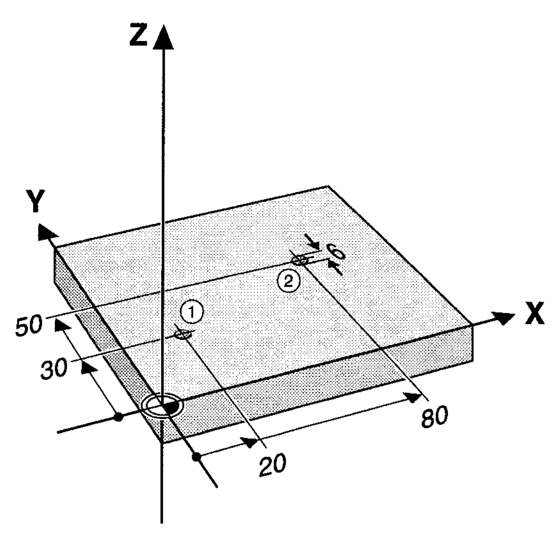
Example: PECKING

Hole coordinates:

① X = 20 mm Y = 30 mm

② X = 80 mm Y = 50 mm

Hole diameter:	6	mm
Setup clearance:	2	mm
Total hole depth:	15	mm
Pecking depth:	10	mm
Dwell time:	1	second
Feed rate:	80	mm/min



PECKING cycle in a part program

```
%S851 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 %S851 G71 *
```

TAPPING with floating tap holder (G84)

Process

- The thread is cut in one pass.
- Once the tool has reached the total hole depth, the direction of spindle rotation is reversed and the tool is retracted to the starting position at the end of the dwell time.
- At the starting position, the direction of spindle rotation reverses once again.

Required tool

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

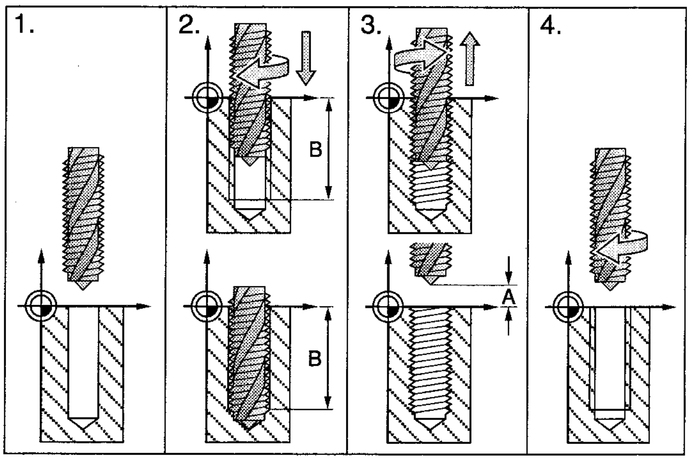


Fig. 8.3: TAPPING cycle

Input data

- SETUP CLEARANCE Ⓐ:
  Distance between tool tip (at starting position) and workpiece surface.
  Standard value: approx. 4 x thread pitch
- TOTAL HOLE DEPTH Ⓑ (thread length):
  Distance between workpiece surface and end of thread.
- DWELL TIME IN SECONDS:
  Enter a dwell time between 0 and 0.5 seconds to avoid wedging of the tool during retraction (further information is available from the machine manufacturer).
- FEED RATE:
  Traversing speed of the tool during tapping.

Calculations

The feed rate is calculated as follows:

$$F = S x p$$

where *F* is the feed rate (mm/min), *S* is the spindle speed (rpm) and *p* is the thread pitch (mm).

- When a cycle is being run, the spindle speed override knob is disabled. The feed rate override knob is only active within a limited range (preset by the machine manufacturer).
- For tapping right-hand threads activate the spindle with M3, for left-hand threads use M4.

Example: Tapping with a floating tap holder

Cutting an M6 thread at 100 rpm

Tapping coordinates:

X = 50 mm Y = 20 mm

Pitch p = 1 mm

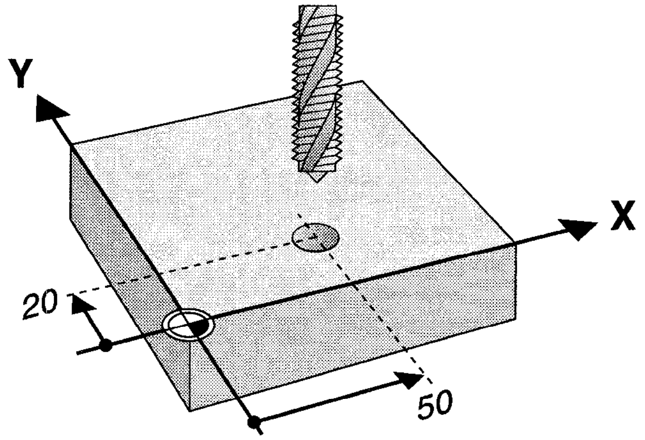
$F = S \times p \Rightarrow F = 100 \cdot 1 = 100$ mm/min

Setup clearance: 3 mm

Thread depth: 20 mm

Dwell time: 0.4 second

Feed rate: 100 mm/min



TAPPING cycle in a part program

```
%S871 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 %S871 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
- Increased traverse range of the spindle axis due to absence of a floating tap holder

The machine and the 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.
- THREAD PITCH Ⓒ:
  The sign differentiates between right-hand and left-hand threads:
  + = right-hand thread
  – = left-hand thread

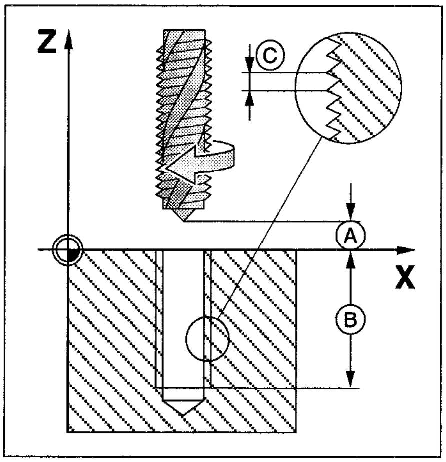


Fig. 8.4: Input data for RIGID TAPPING cycle

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.

SLOT MILLING (G74)

Process

Roughing process:

- The tool penetrates the workpiece from the starting position, offset by the oversize, then mills in the longitudinal direction of the slot.
- The oversize is calculated as: (slot width – tool diameter) / 2.
- After downfeed at the end of the slot, milling is performed in the opposite direction.
  This process is repeated until the programmed milling depth is reached.

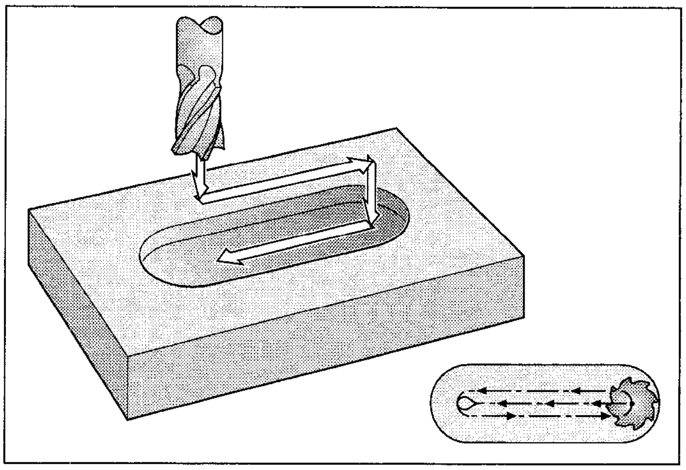


Fig. 8.5: SLOT MILLING cycle

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.

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.

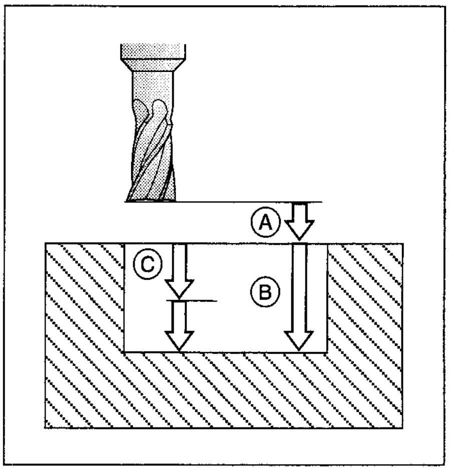


Fig. 8.6: Infeeds and distances for the SLOT MILLING cycle

Input data

- SETUP CLEARANCE Ⓐ
- MILLING DEPTHⒷ: Slot depth.
- 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.

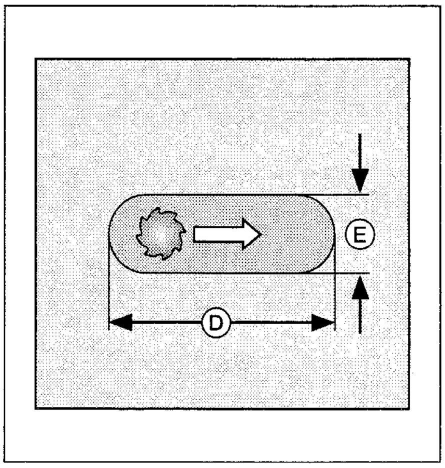


Fig. 8.7: Side lengths of the slot

Example: Slot milling

A horizontal slot (50 mm x 10 mm) and a vertical slot (80 mm x 10 mm) are to be milled.

The tool radius in the length direction of the slot is taken into account for the starting position.

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

Starting position, slot ②:
X = 20 mm Y = 14 mm

SLOT DEPTH: 15 mm

Setup clearance: 2 mm

Milling depth: 15 mm

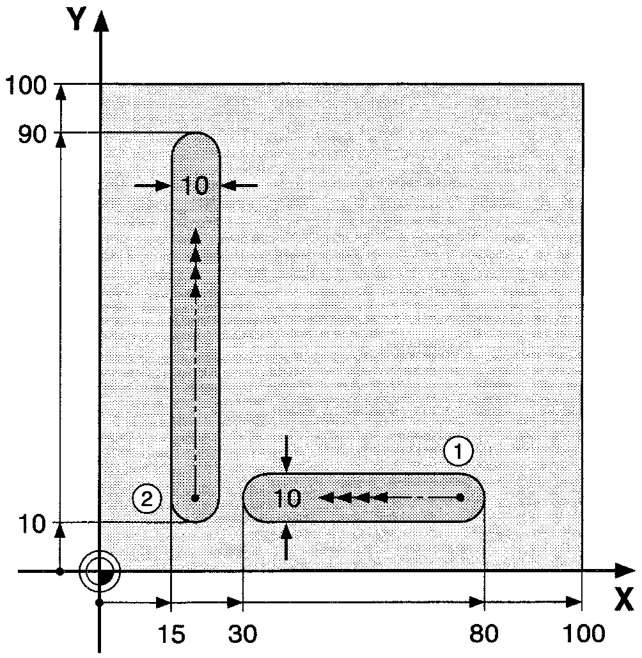
Pecking depth: 5 mm

Feed rate for pecking: 80 mm/min

	①	②
Slot length	50 mm	80 mm
1st milling direction	–	+

Slot width: 10 mm

Feed rate: 120 mm/min



SLOT MILLING cycle in a part program

```
%S810I G71 * ......................................................... Start of program
N10 G30 G17 X+0 Y+0 Z–20 * .................................. Define workpiece blank
N20 G31 G90 X+100 Y+100 Z+0 *
N30 G99 T1 L+0 R+4 * ........................................... Define tool
N40 T1 G17 S2000 * .............................................. Call tool
N50 G74 P01 2 P02 –15 P03 5 P04 80 P05 X–50
P06 Y+10 P07 120 * ................................................ Define slot parallel to X axis
N60 G00 G40 G90 Z+100 M06 * ............................. Retract in the infeed axis, insert tool
N70 X+76 Y+15 M03 * ............................................ Approach starting position, spindle ON
N80 Z+2 M99 * ......................................................... Pre-position in Z to setup clearance, cycle call ①
N90 G74 P01 2 P02 –15 P03 5 P04 80 P05 Y+80
P06 X+10 P07 120 * ................................................ Define slot parallel to Y axis
N100 X+20 Y+14 M99 * ......................................... Approach starting position, cycle call ②
N110 Z+100 M02 * ................................................. Retract in the infeed axis, end of program
N99999 %S810I G71 *
```

POCKET MILLING (G75/G76)

Process

The rectangular pocket milling cycle is a roughing cycle, in which

- the tool penetrates the workpiece at the starting position (pocket center)
- the tool subsequently follows the programmed path at the specified feed rate (see Figure 8.10)

The cutter begins milling in the positive direction of the axis of the longer side. The cutter always starts in the positive Y direction on square pockets. At the end of the cycle, the tool is retracted to the starting position.

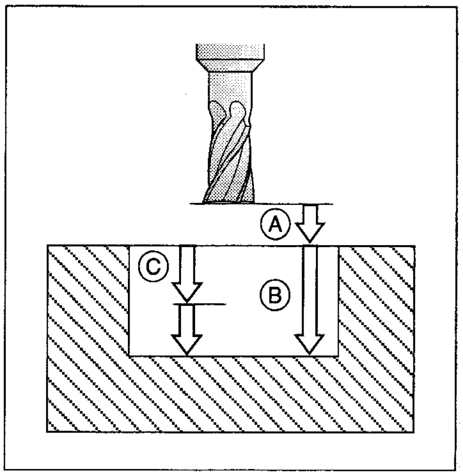


Fig. 8.8: Infeeds and distances for the POCKET MILLING cycle

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

Input data

- SETUP CLEARANCE Ⓐ
- MILLING DEPTH Ⓑ
- 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.

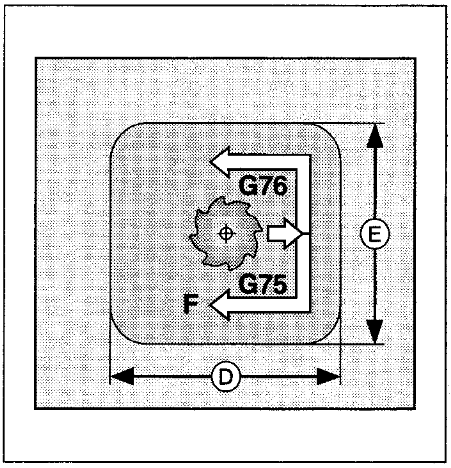


Fig. 8.9: Side lengths of the pocket

Calculations

The stepover factor *k* is calculated as follows:

$$k = K \times R$$

where *K* is the overlap factor (preset by the machine manufacturer) and *R* is the cutter radius.

Corner radius

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

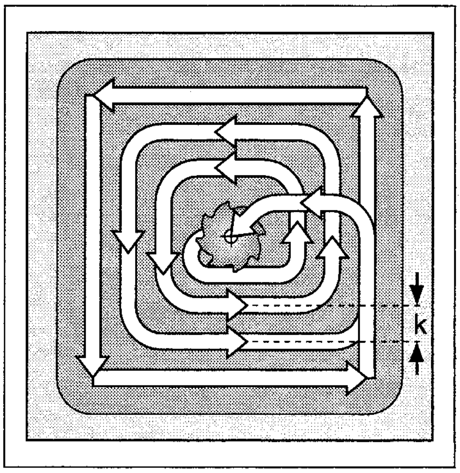


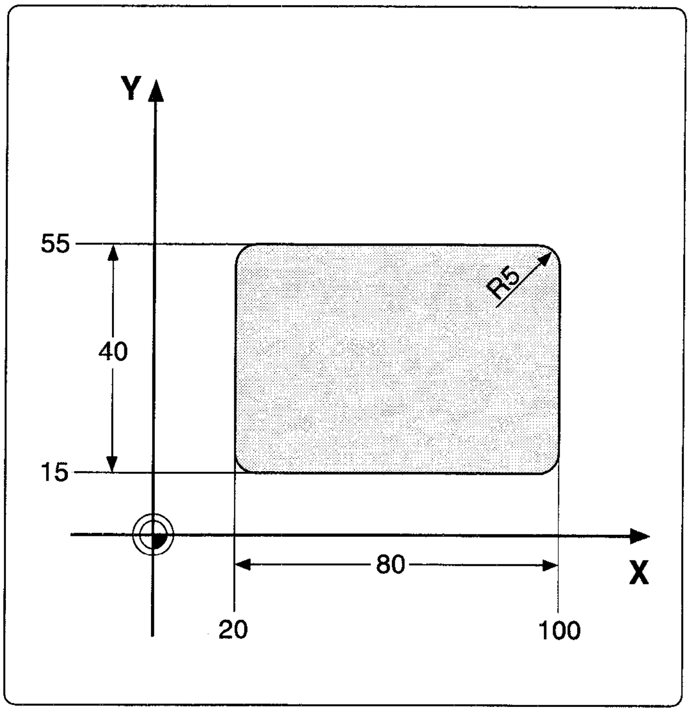
Fig. 8.10: Tool path for roughing-out

Example: Rectangular pocket milling

Pocket center coordinates:

X = 60 mm Y = 35 mm

Setup clearance:	2	mm
Milling depth:	10	mm
Pecking depth:	4	mm
Feed rate for pecking:	80	mm/min
First side length:	80	mm
Second side length:	40	mm
Milling feed rate:	100	mm/min
Direction of cutter path:	+	



POCKET MILLING cycle in a part program

```
%S812I G71 * ........................................................... Start of program
N10 G30 G17 X+0 Y+0 Z–20 * .................................. Define workpiece blank
N20 G31 G90 X+110 Y+100 Z+0 *
N30 G99 T1 L+0 R+5 * .............................................. Define tool
N40 T1 G17 S2000 * ................................................. Call tool
N50 G76 P01 2 P02 –10 P03 4 P04 80 P05 X+80
P06 Y+40 P07 100 * .................................................. Define POCKET MILLING cycle
N60 G00 G40 G90 Z+100 M06 * ............................... Retract in the infeed axis, insert tool
N70 X+60 Y+35 M03 * .............................................. Approach the starting position (center of pocket), spindle ON
N80 Z+2 M99 * ......................................................... Pre-position in Z to setup clearance, cycle call
N90 Z+100 M02 * ...................................................... Retract in the infeed axis, end of program
N99999 %S812I G71 *
```

CIRCULAR POCKET MILLING (G77/G78)

Process

- Circular pocket milling is a roughing cycle in which the tool penetrates the workpiece from the starting position (pocket center).
- The cutter subsequently follows a spiral path (shown in Figure 8.11) at the programmed feed rate. The stepover factor is determined by the value *k* (see G75/G76 POCKET MILLING, Calculations).
- The process is repeated until the programmed milling depth is reached.
- At the end of the cycle, the tool is retracted to the starting position.

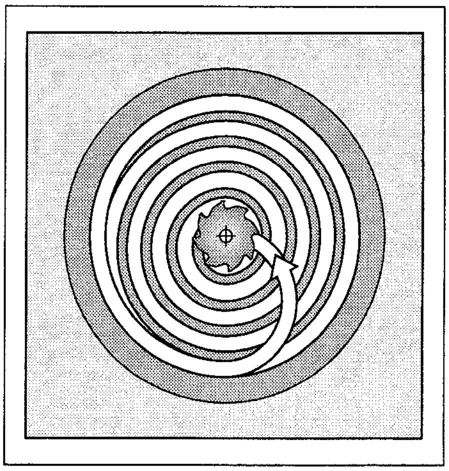


Fig. 8.11: Cutter path for roughing-out

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
- 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

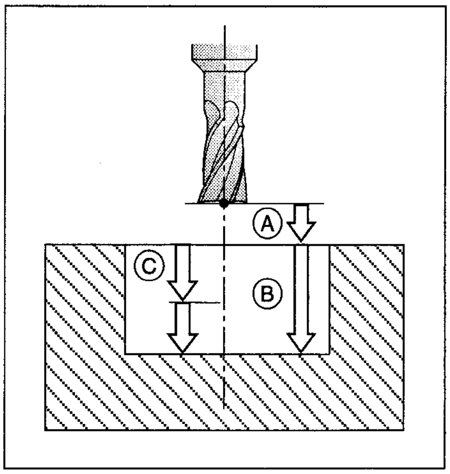


Fig. 8.12: Distances and infeeds for CIRCULAR POCKET MILLING

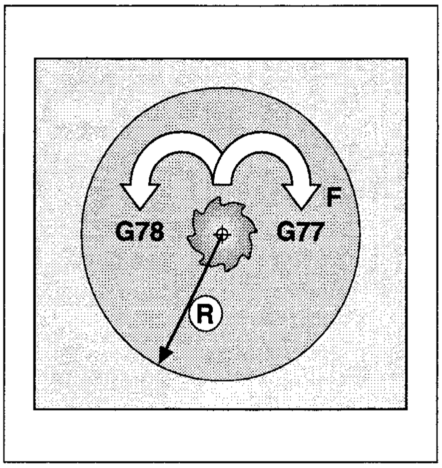


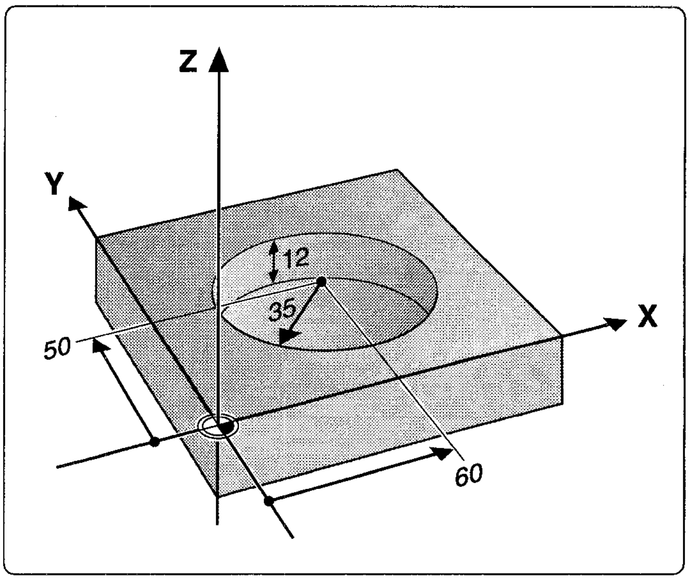
Fig. 8.13: Direction of the cutter path

Example: Milling a circular pocket

Pocket center coordinates:

X = 60 mm Y = 50 mm

Setup clearance:	2	mm
Milling depth:	12	mm
Pecking depth:	6	mm
Feed rate for pecking:	80	mm/min
Circle radius:	35	mm
Milling feed rate:	100	mm/min
Direction of the cutter path:	–	



CIRCULAR POCKET cycle in a part program

```
%S814I G71 * .......................................... Start of program
N10 G30 G17 X+0 Y+0 Z–20 * ............................ Define workpiece blank
N20 G31 G90 X+100 Y+100 Z+0 *
N30 G99 T1 L+0 R+4 * .................................. Define tool
N40 T1 G17 S2000 * .................................... Call tool
N50 G77 P01 2 P02 –12 P03 6 P04 80 P05 35
P06 100 * ............................................. Define circular pocket milling cycle
N60 G00 G40 G90 Z+100 M06 * ........................... Retract in the infeed axis, insert tool
N70 X+60 Y+50 M03 * ................................... Approach the starting position (center of pocket), spindle ON
N80 Z+2 M99 * ......................................... Pre-position in Z to setup clearance, cycle call
N90 Z+100 M02 * ....................................... Retract in the infeed axis, end of program
N99999 %S814I G71 *
```

8.3 SL Cycles

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)

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 deduces from these data whether the specific subprogram describes a pocket or an island:

- The control recognizes a *pocket* if the tool path lies *inside* the contour
- The control recognizes an *island* if the tool path lies *outside* the contour

- The machining of the SL contour is determined by MP 7420.
- It is a good idea to run a graphic simulation before executing a program to see whether the contours were correctly defined.
- The memory capacity for SL cycles is limited. One SL cycle can contain, for example, a maximum of 128 straight line blocks.
- All coordinate transformations are available when 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. Subsequently entered coordinate axes will be ignored.

CONTOUR GEOMETRY (G37)

Application

All subprograms that are superimposed to define the contour are listed in Cycle G37 CONTOUR GEOMETRY.

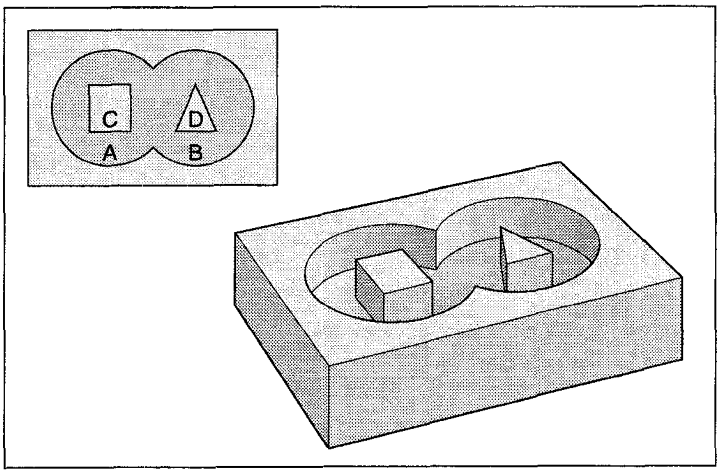


Fig. 8.14: Example of an SL contour. *A* and *B* are pockets, *C* and *D* are islands

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.

Input data

- SETUP CLEARANCE Ⓐ
- MILLING DEPTH Ⓑ
- PECKING DEPTH Ⓒ
- FEED RATE FOR PECKING:
  Traversing speed of the tool during penetration
- FINISHING ALLOWANCE Ⓓ:
  Allowance in the machining plane (positive value)
- ROUGH-OUT ANGLE Ⓐ:
  Feed direction for roughing-out.
  The rough-out angle is relative to the angle reference axis and can be set, so that the resulting cuts are as long as possible with few cutting movements.
- FEED RATE:
  Traversing speed of the tool in the machining plane

The machine parameters determine whether

- the contour is milled first and then surface machined, or vice versa
- the contour is milled conventionally or by climb cutting
- all pockets are roughed-out first and then contour-milled over all infeeds, or whether
- contour milling and roughing-out are performed mutually for each infeed

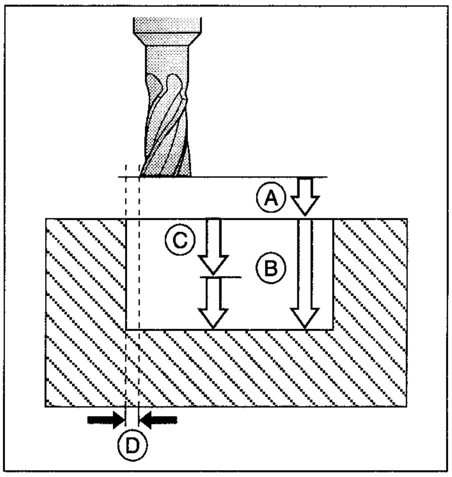


Fig. 8.15: Infeeds and distances of the ROUGH-OUT cycle

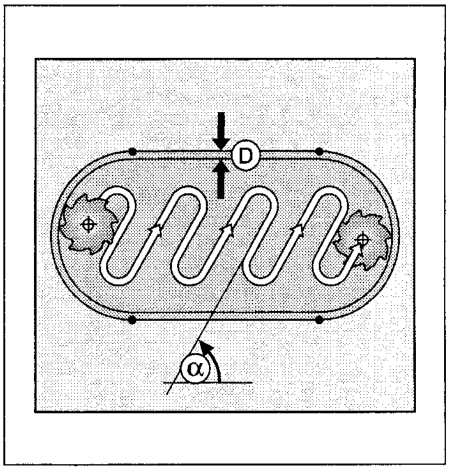


Fig. 8.16: Cutter path for roughing-out

Example: Roughing-out a rectangular 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

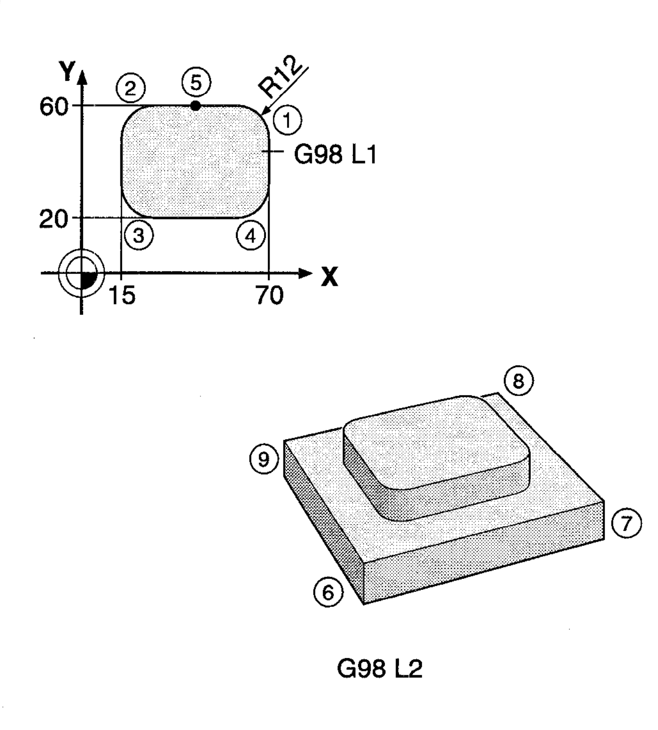
Coordinates of the auxiliary pocket:

	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



ROUGH-OUT cycle in a part program

```
%S818I G71 * ........................................................Start of program
N10 G30 G17 X+0 Y+0 Z–20 * ..................................Define workpiece blank
N20 G31 X+100 Y+100 Z+0 *
N30 G99 T1 L+0 R+3 * ...........................................Define tool
N40 T1 G17 S2500 * ................................................Call tool
N50 G37 P01 2 P02 1 * ..........................................In the CONTOUR GEOMETRY cycle, state that the contour
                                                                     elements are described in subprograms 2 and 1
N60 G57 P01 2 P02 –15 P03 8 P04 100 P05 +0
P06 +0 P07 500 * ....................................................Cycle definition ROUGH-OUT
N70 G00 G40 G90 Z+100 M06 * .............................Retract in the infeed axis, insert tool
N80 X+40 Y+50 M03 * ............................................Pre-position in X/Y, spindle ON
N90 Z+2 M99 * .......................................................Pre-position in Z to setup clearance, cycle call
N100 Z+100 M02 *

N110 G98 L1 *                                                        Subprogram 1:
N120 G01 G42 X+40 Y+60 *                                             Geometry of the island
N130 X+15 *                                                          (radius compensation G42 and machining in counterclockwise
                                                                     direction: the contour element is an island)
N150 Y+20 *
N160 G25 R12 *
N170 X+70 *
N180 G25 R12 *
N190 Y+60 *
N200 G25 R12 *
N210 X+40 *
N220 G98 L0 *

N230 G98 L2 *                                                        Subprogram 2:
N240 G01 G41 X–5 Y–5 *                                               Geometry of the auxiliary pocket:
N250 X+105 *                                                         External boundary of the area to
N260 Y+105 *                                                         be machined
N270 X–5 *                                                           (radius compensation G41 and machining in counterclockwise
N280 Y–5 *                                                           direction: the contour element is a pocket)
N290 G98 L0 *
N99999 %S818I G71 *
```

Overlapping contours

Pockets and islands can also be overlapped to form a new contour. The area of a pocket can thus be enlarged by another pocket or reduced by an island.

Starting position

Machining begins at the starting position of the first pocket listed in Cycle G37 CONTOUR GEOMETRY. The starting position should be located as far as possible from the superimposed contours.

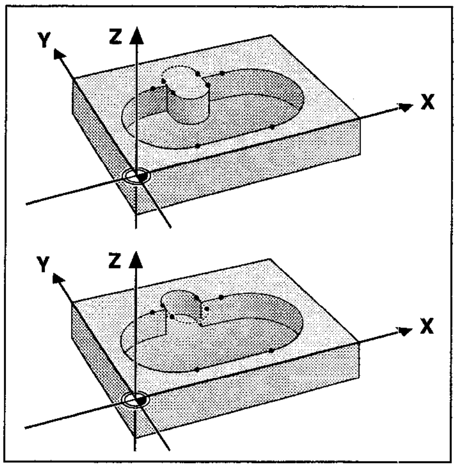


Fig. 8.17: Examples of overlapping contours

Example: Overlapping pockets

The machining process starts with the first contour label defined in block 6. The first pocket must begin outside the second pocket.

Inside machining with a center-cut end mill (ISO 1641), tool radius 3 mm

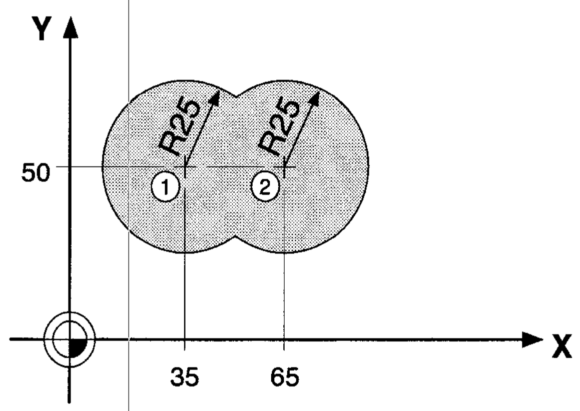
Coordinates of the circle centers:

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

Circle radii

R = 25 mm

Setup clearance:	2	mm
Milling depth:	10	mm
Pecking depth:	5	mm
Feed rate for pecking:	500	mm/min
Finishing allowance:	0	
Rough-out angle:	0	
Milling feed rate:	500	mm/min



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

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 *          
N130    I+35 J+50 G03 X+10 Y+50 *    }  A   Left pocket
N140    G98 L0 *                     

N150    G98 L2 *                     
N160    G01 G41 X+90 Y+50 *          
N170    I+65 J+50 G03 X+90 Y+50 *    }  B   Right pocket
N180    G98 L0 *                     
N99999  % S820I G71 *                
```

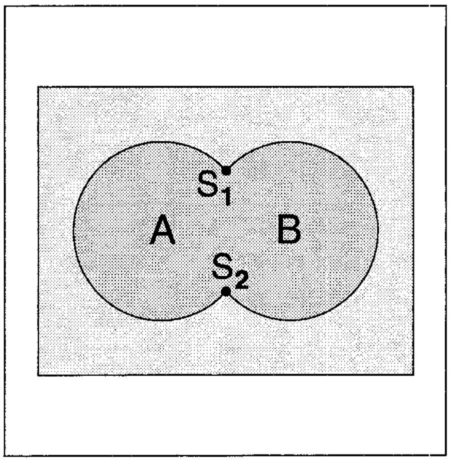


Fig. 8.18: Points of intersection S_1 and S_2 of pockets *A* and *B*

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

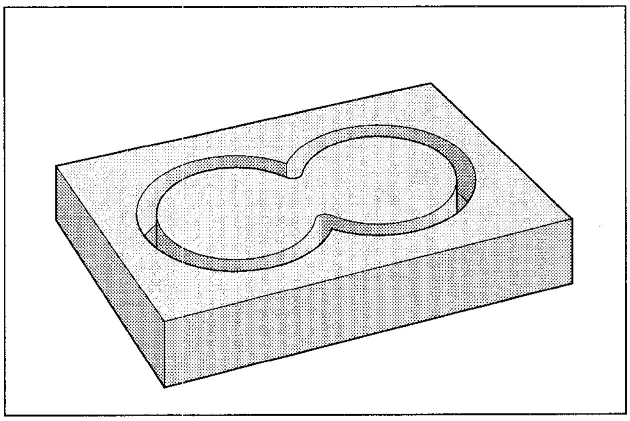


Fig. 8.19: Outline is machined first

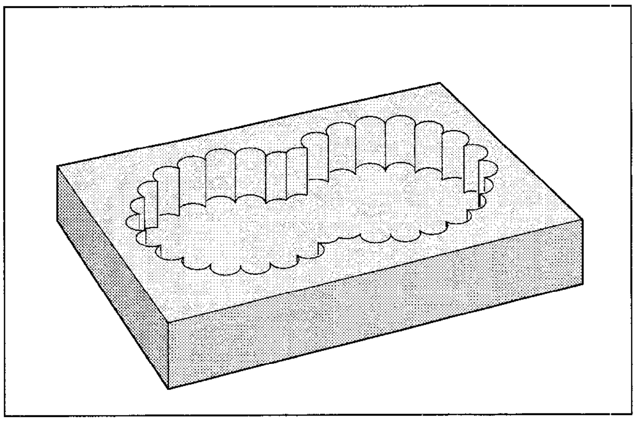


Fig. 8.20: Surface is machined first

Area of inclusion

Both surfaces *A* and *B* are to be machined, including the mutually overlapped area.

- *A* and *B* must be pockets.
- The first pocket (in Cycle G37) must start outside the second pocket.

```
N110   G98 L1 *
N120   G01 G41 X+10 Y+50 *
N130   I+35 J+50 G03 X+10 Y+50 *
N140   G98 L0 *

N150   G98 L2 *
N160   G01 G41 X+90 Y+50 *
N170   I+65 J+50 G03 X+50 Y+50 *
N180   G98 L0 *
```

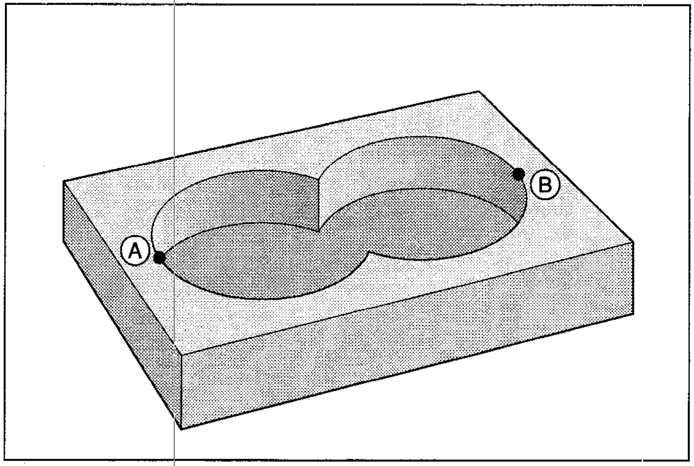


Fig. 8.21: Overlapping pockets: area of inclusion

Area of exclusion

Surface *A* is to be machined without the portion overlapped by *B*.

- *A* must be a pocket and *B* an island.
- *A* must start outside of *B*.

```
N110   G98 L1 *
N120   G01 G41 X+10 Y+50 *
N130   I+35 J+50 G03 X+10 Y+50 *
N140   G98 L0 *

N150   G98 L2 *
N160   G01 G42 X+90 Y+50 *
N170   I+65 J+50 G03 X+90 Y+50 *
N180   G98 L0 *
```

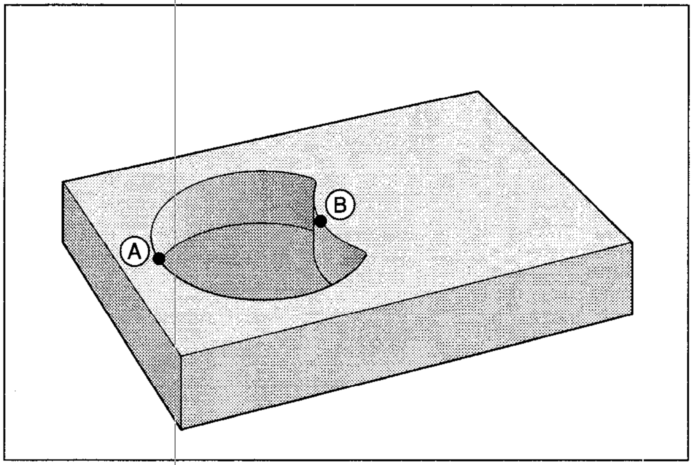


Fig. 8.22: Overlapping pockets: area of exclusion

Area of intersection

Only the area covered by both *A* and *B* is to be machined.

- *A* and *B* must be pockets.
- *A* must start inside *B*.

```
N110   G98 L1 *
N120   G01 G41 X+60 Y+50 *
N130   I+35 J+50 G03 X+60 Y+50 *
N140   G98 L0 *

N150   G98 L2 *
N160   G01 G41 X+90 Y+50 *
N170   I+65 J+50 G03 X+90 Y+50 *
N180   G98 L0 *
```

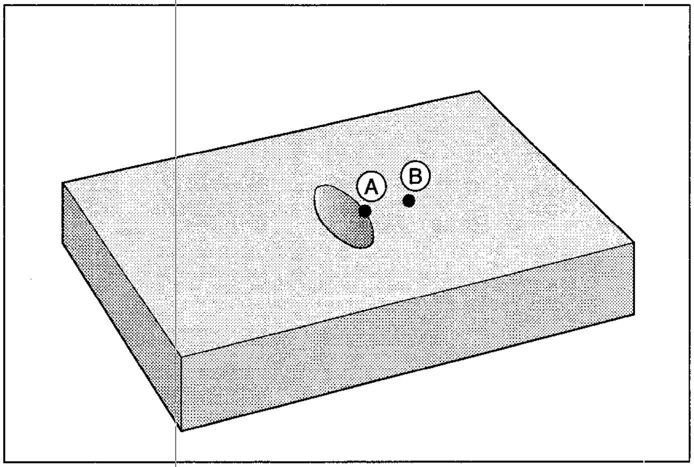


Fig. 8.23: Overlapping pockets: area of intersection

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

Subprogram: Overlapping islands

An island always requires an additional boundary, the pocket (here, G98 L1).

```
%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.

```
N180   G98 L2 *
N190   G01 G42 X+10 Y+50 *
N200   I+35 Y+50 G03 X+10 Y+50 *
N210   G98 L0 *
N220   G98 L3 *
N230   G01 G42 X+90 Y+50 *
N240   I+65 J+50 G03 X+90 Y+50 *
N250   G98 L0 *
N99999 % S822 I G71
```

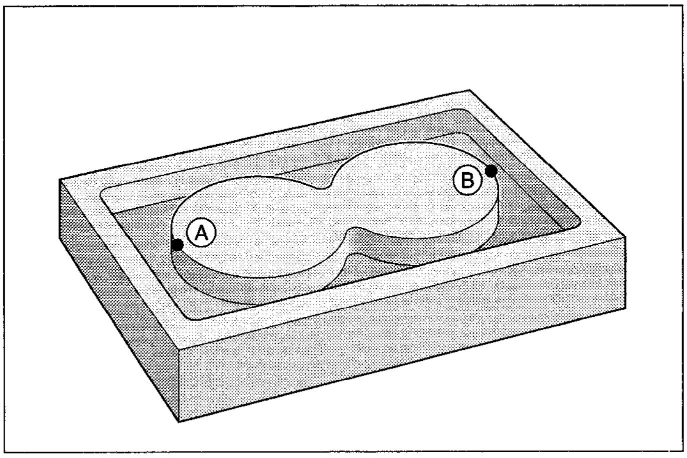


Fig. 8.24: Overlapping islands: area of inclusion

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

Area of exclusion

Surface *A* is to be left unmachined except for 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 S8221 G71*
```

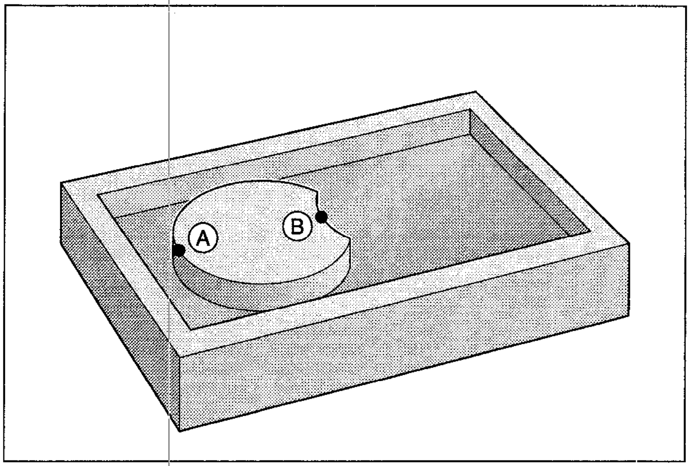


Fig. 8.25: Overlapping islands: area of exclusion

Area of intersection

Only the area common to *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 % S8221 G71
```

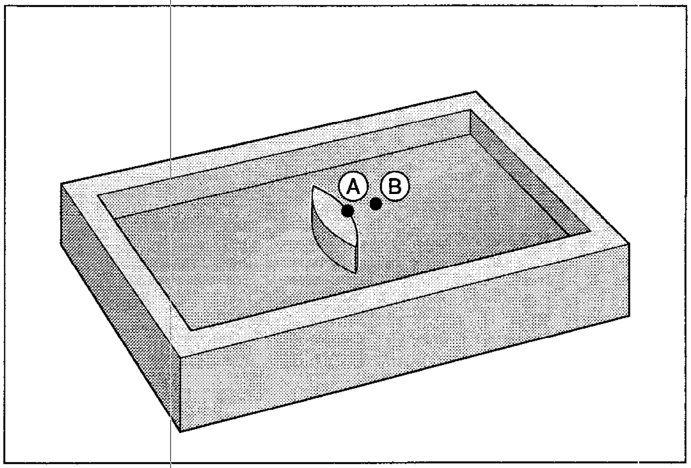


Fig. 8.26: Overlapping islands: area of intersection

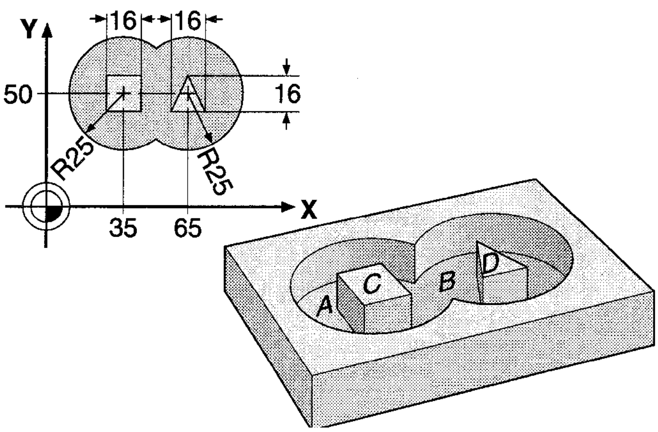
Example: Overlapping pockets and islands

PGM S824I is similar to PGM S820I but adds the islands *C* and *D*.

Tool: Center-cut end mill (ISO 1641), radius 3 mm

The contour is composed of the following elements:

Two overlapping pockets *(A* and *B)*, and two islands within the pockets *(C* and *D)*.



Cycle in a part program

```
%S824I G71 *
N10 G30 G17 X+0 Y+0 Z-20 *
N20 G31 X+100 Y+100 Z+0 *
N30 G99 T1 L+0 R+3 *
N40 T1 G17 S2500 *
N50 G37 P01 1 P02 2 P03 3 P04 4 *
N60 G57 P01 2 P02 -10 P03 5 P04 100 P05 +2 P06 +0 P07 500 *
N70 G00 G40 G90 Z+100 M06 *
N80 X+50 Y+50 M03 *
N90 Z+2 M99 *
N100 Z+100 M02 *

N110 G98 L1 *
N120 G01 G41 X+10 Y+50 *
N130 I+35 J+50 G03 X+10 Y+50 *
N140 G98 L0 *

N150 G98 L2 *
N160 G01 G41 X+90 Y+50 *
N170 I+65 J+50 G03 X+90 Y+50 *
N180 G98 L0 *

N190 G98 L3 *
N200 G01 G41 X+27 Y+42 *
N210 Y+58 *
N220 X+43 *
N230 Y+42 *
N240 X+27 *
N250 G98 L0 *

N260 G98 L4 *
N270 G01 G42 X+57 Y+42 *
N280 X+73 *
N290 X+65 Y+58 *
N300 X+57 Y+42 *
N310 G98 L0 *
N99999 %S824I G71 *
```

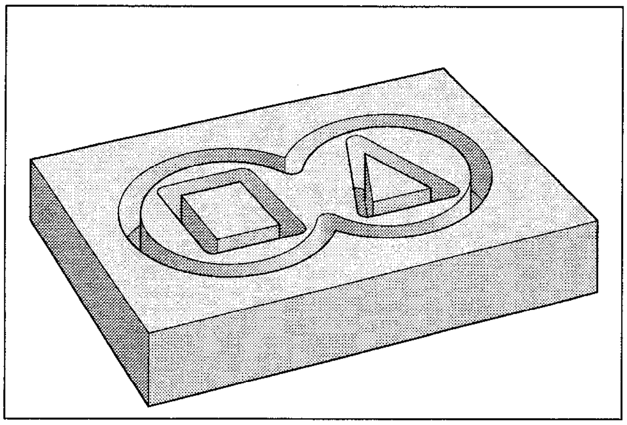


Fig. 8.27: Milling of outline

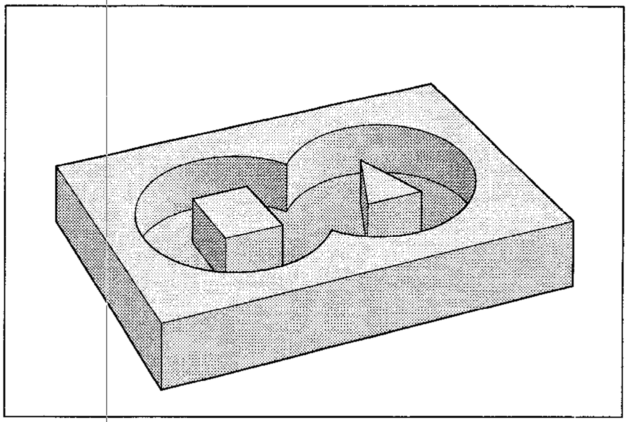


Fig. 8.28: Finished workpiece

PILOT DRILLING (G56)

This cycle performs pilot drilling of holes for cutter infeed at the starting points of the subcontours. With SL contours consisting of several overlapping pockets and islands, the cutter infeed point is the starting point of the first subcontour:

- The tool is positioned at setup clearance over the first infeed point.
- The drilling sequence is identical to fixed Cycle G83 PECKING.
- The tool is then positioned above the second infeed point, and the drilling process is repeated.

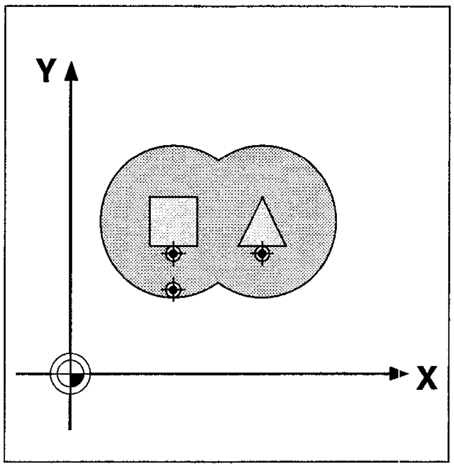


Fig. 8.29: Example of cutter infeed points for PECKING

Input data

- SETUP CLEARANCE
- TOTAL HOLE DEPTH
- PECKING DEPTH
- FEED RATE

} identical to Cycle G83 PECKING

- FINISHING ALLOWANCE Ⓓ
  Allowed material for the drilling operation (see Figure 8.30).
  The sum of the tool radius and the finishing allowance should be the same for pilot drilling as for roughing out.

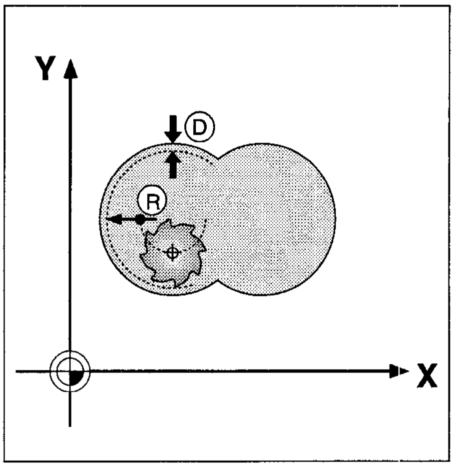


Fig. 8.30: Finishing allowance

CONTOUR MILLING (G58/G59)

The CONTOUR MILLING cycles are used to finish-mill the contour pocket. The cycles can also be used generally for milling contours.

Sequence

- The tool is positioned at setup clearance over the first starting point.
- Moving at the programmed feed rate, the tool then penetrates to the first pecking depth.
- Upon reaching the first pecking depth, the tool mills the first contour at the programmed feed rate in the specified direction of rotation.
- At the infeed point, the control advances the tool to the next pecking depth.

This process is repeated until the programmed milling depth is reached. The remaining subcontours are milled in the same manner.

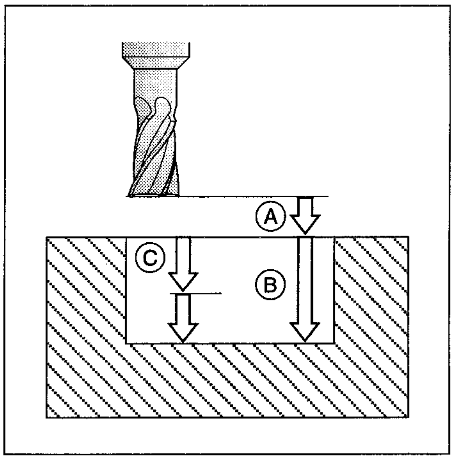


Fig. 8.31: Infeeds and distances for CONTOUR MILLING

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

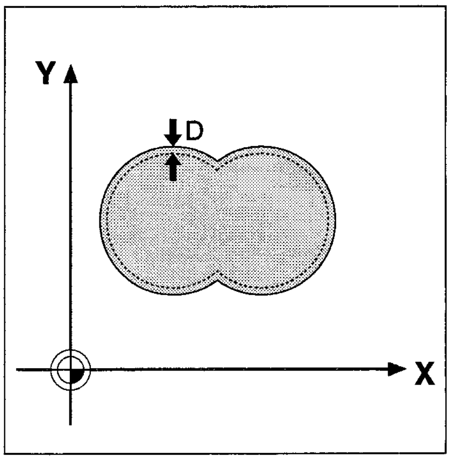


Fig. 8.32: Finishing allowance

Input data

- SETUP CLEARANCE Ⓐ
- MILLING DEPTH Ⓑ
- 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 the cycles PILOT DRILLING, ROUGH-OUT and CONTOUR MILLING in part programming.

1. **List of contour subprograms**

   G37
   No call

2. **Drilling**

   Define and call the drilling tool
   G56
   Pre-positioning
   Cycle call

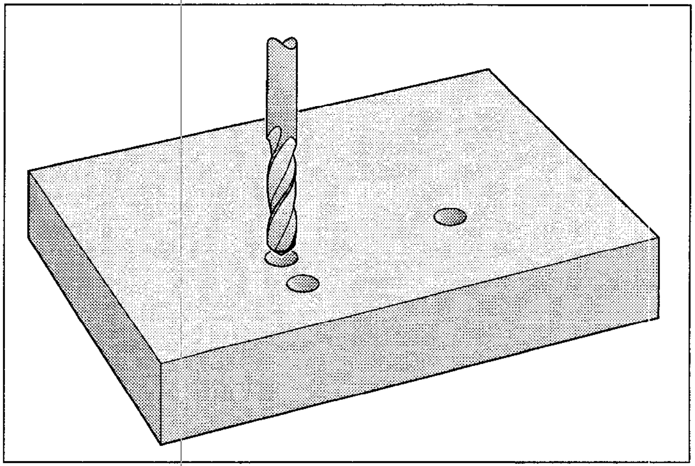


Fig. 8.33: PILOT DRILLING cycle

3. **Rough-out**

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

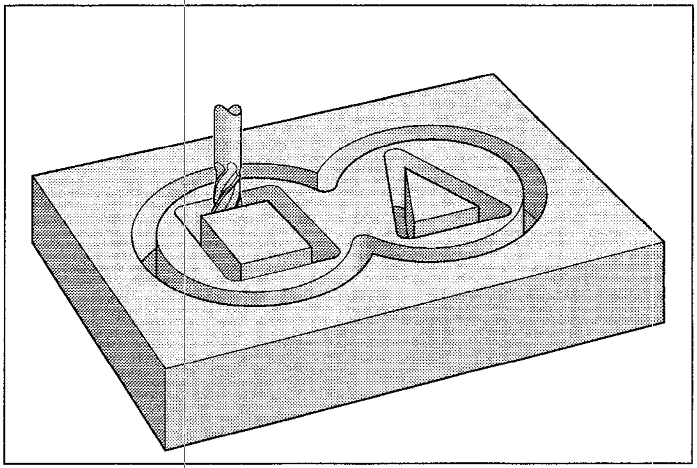


Fig. 8.34: ROUGH-OUT cycle

4. **Finishing**

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

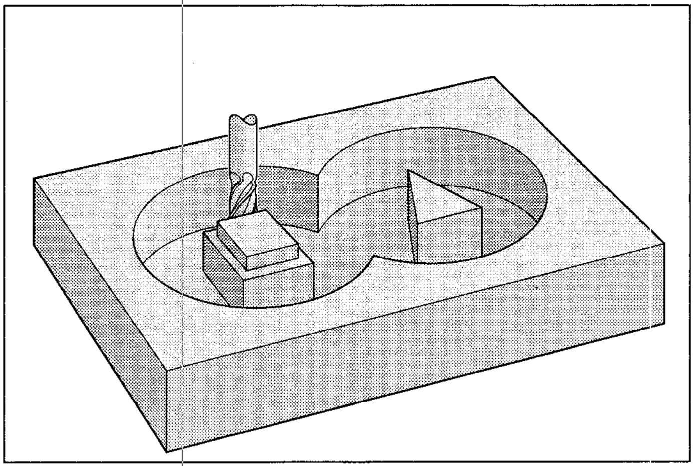


Fig. 8.35: CONTOUR MILLING cycle

5. **Contour subprograms**

   M02 *
   Subprograms for the subcontours

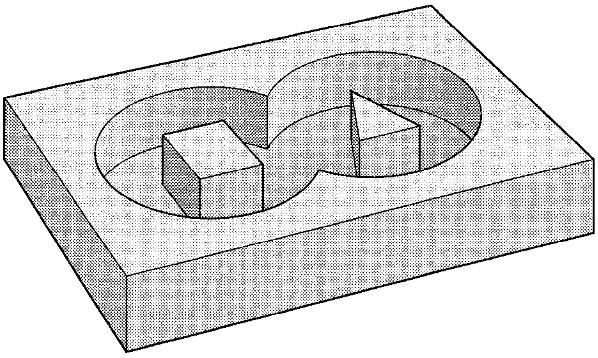
Example: Overlapping pockets with islands

Inside machining with pilot drilling, roughing-out and finishing.

PGM S829I is based on S824I:

The main program section has been expanded by the cycle definitions and calls for pilot drilling and finishing.

The contour subprograms 1 to 4 are identical to the ones in PGM S824I (see page 8-24) and are to be added after block N300.



```
%S829I G71 * ..................................... Start of program
N10 G30 G17 X+0 Y+0 Z–20 * ....................... Define workpiece blank
N20 G31 X+100 Y+100 Z+0 *
N30 G99 T1 L+0 R+2.5 * ........................... Tool definition: drill bit
N40 G99 T2 L+0 R+3 * ............................. Tool definition: roughing mill
N50 G99 T3 L+0 R+2.5 * ........................... Tool definition: finishing mill
N60 L10,0 * ...................................... Subprogram call for tool change
N70 G38 M06 * .................................... Program STOP
N80 T1 G17 S2500 * ............................... Tool call: drill bit
N90 G37 P01 1 P02 2 P03 3 P04 4 * ................ Cycle definition: Contour Geometry
N100 G56 P01 2 P02 –10 P03 5 P04 500 P05 +2 * .... Cycle definition: Pilot Drilling
N110 Z+2 M03 *
N120 G79 * ....................................... Cycle call: Pilot Drilling
N130 L10,0 *
N140 G38 M06 * ................................... Tool change
N150 T2 G17 S1750 * .............................. Tool call: roughing mill
N160 G57 P01 2 P02 –10 P03 5 P04 100 P05+2
P06+0 P07 500 * .................................. Cycle definition: Rough-Out
N170 Z+2 M03 *
N180 G79 * ....................................... Cycle call: Rough-Out
N190 L10,0 *
N200 G38 M06 * ................................... Tool change
N210 T3 G17 S2500 * .............................. Tool call: finishing mill
N220 G58 P01 2 P02 –10 P03 10 P04 100
P05 500 * ........................................ Cycle definition: Contour Milling
N230 Z+2 M03 *
N240 G79 * ....................................... Cycle call: Contour Milling
N250 Z+100 M02 *

N260 G98 L10 * ................................... Subprogram for tool change
N270 T0 G17 *
N280 G00 G40 G90 Z+100 *
N290 X–20 Y–20 *
N300 G98 L0 *
```

From block N310: Add subprograms on page 8-24

```
N99999 %S829I G71 *
```

8.4 Coordinate Transformations

Coordinate transformations enable a programmed contour to be changed in its location, orientation or size:

- 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 program section.

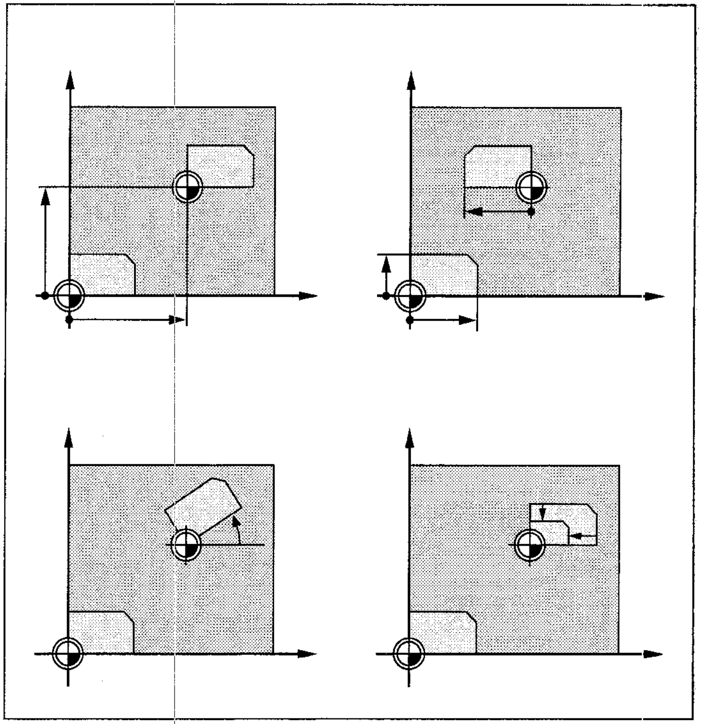


Fig. 8.36: Examples of coordinate transformations

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)

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.

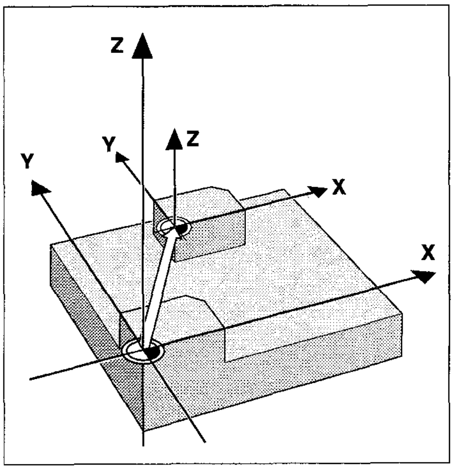


Fig. 8.37: Activation of datum shift

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).

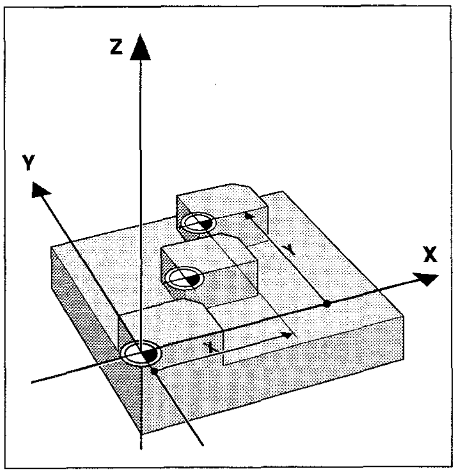


Fig. 8.38: Datum shift, absolute

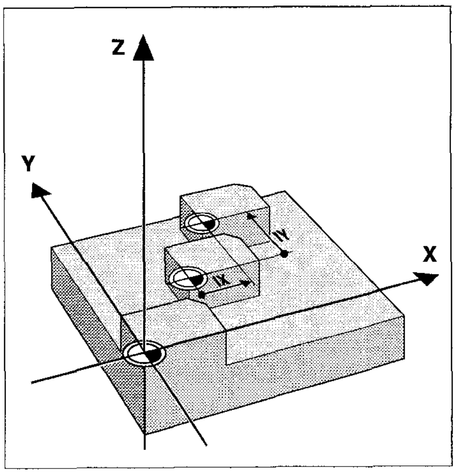


Fig. 8.39: Datum shift, incremental

Cancellation

To cancel a datum shift, enter the datum shift coordinates X = 0, Y = 0 and Z = 0.

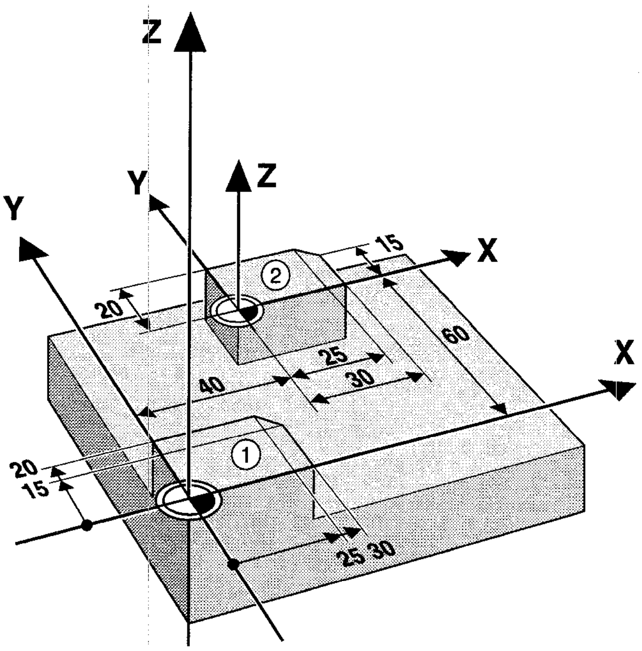
When combining transformations, a datum shift must be programmed before the other transformations.

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

MIRROR IMAGE (G28)

Application

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

Function

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 "flips" over the datum.
- If the datum is located *outside* the contour to be mirrored, the part also "jumps" to another location.

Input data

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

Cancellation

This cycle is cancelled by entering G28 without an axis.

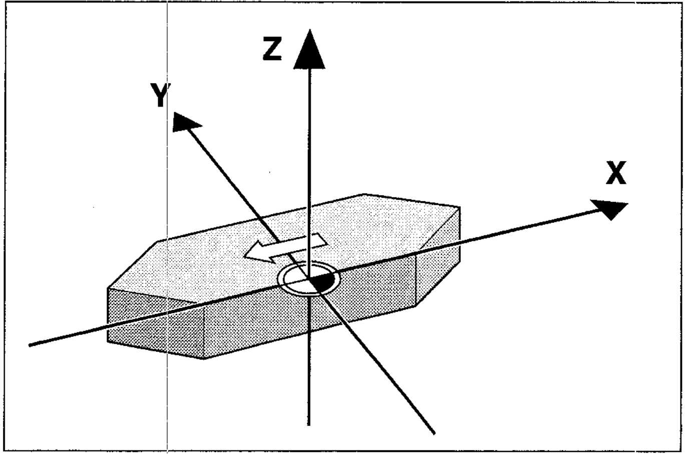


Fig. 8.40: Mirroring a contour

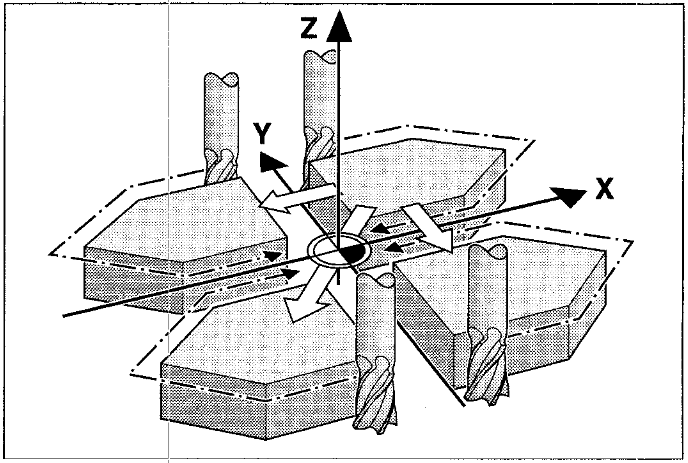


Fig. 8.41: Repeated mirroring, machining direction

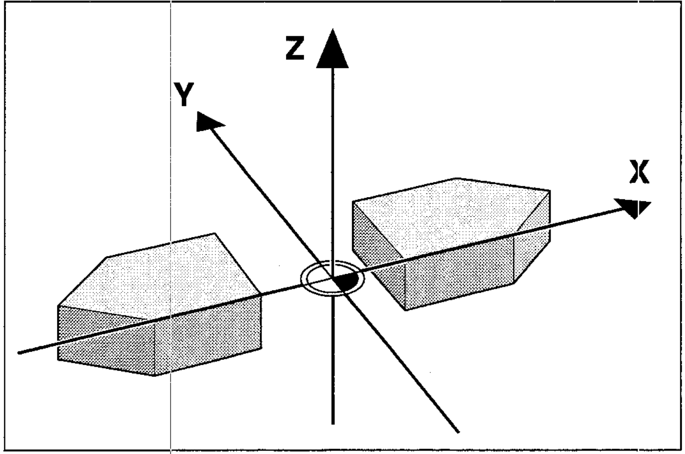
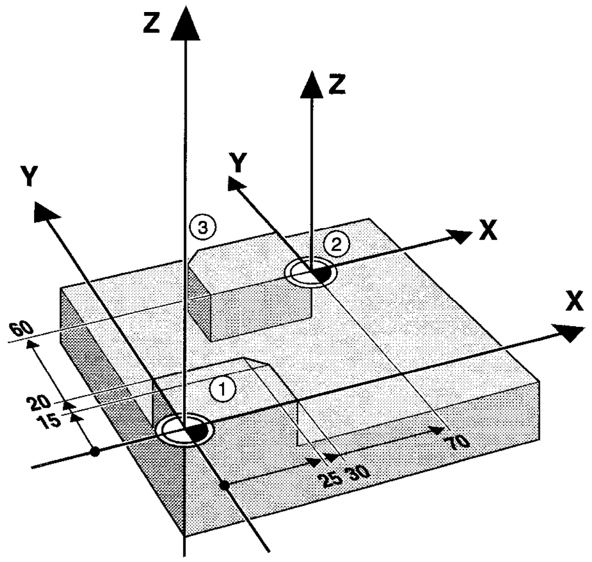


Fig. 8.42: Datum located outside the contour to be mirrored

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 8-32
  .                                           ⎪
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.

Function

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

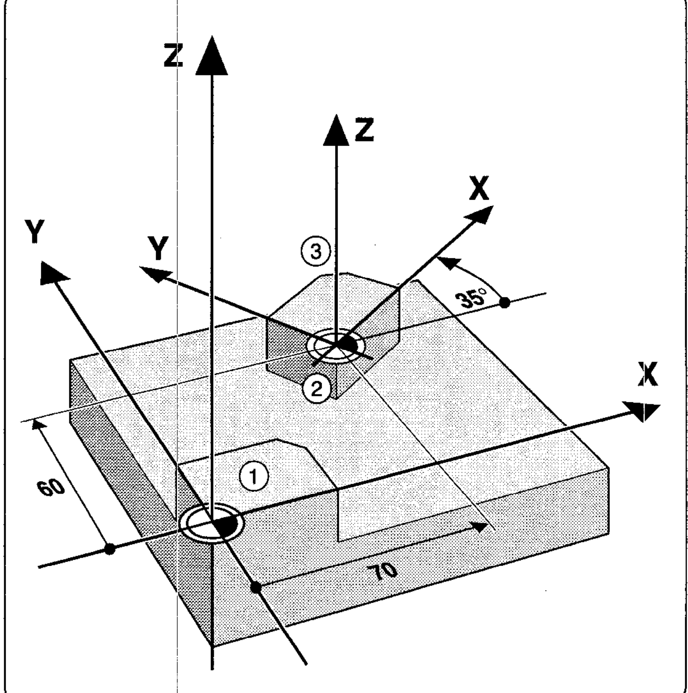
The rotation angle is entered in degrees (°).
Input range: –360° to +360° (absolute or incremental).

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 8-32
     .                                 ⎪
N250 G98 L0 *                          ⎭
N99999 %S846I G71 *
```

The corresponding subprogram (see page 8-32) 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.

Function

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 MP7410)
- 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

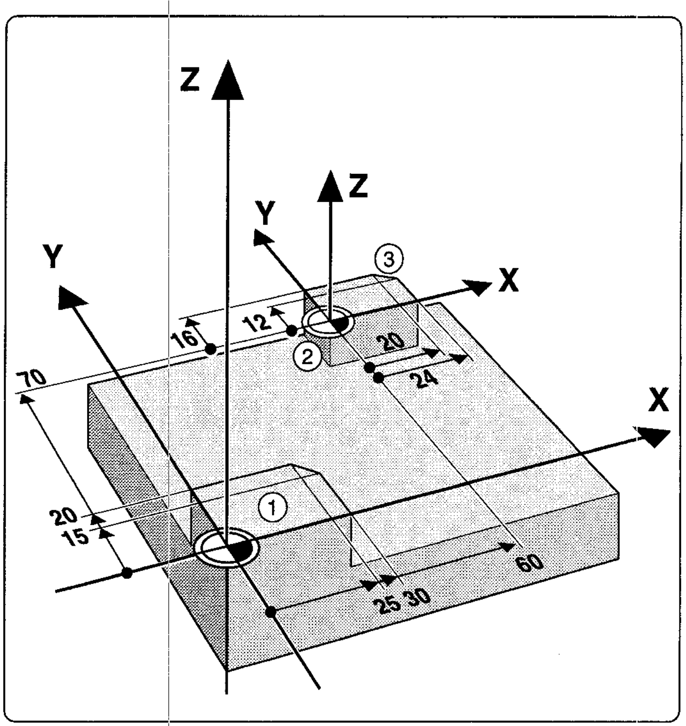
Cancel the scaling factor by entering a scaling factor of 1 in the SCALING FACTOR cycle.

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 8-32
     .                                          |
N250 G98 L0 *                                   )
N99999 %S847I G71 *
```

The corresponding subprogram (see page 8-32) is programmed after M2.

8.5 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.

Function

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 seconds (approx. 8.3 hours) in increments of 0.001 second.

*Resulting NC block: N135 G04 F3**

PROGRAM CALL (G39)

Application and function

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)

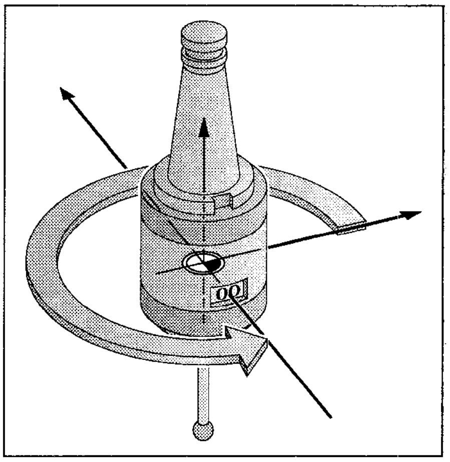


Fig. 8.43: Oriented spindle stop

Application

The control can address the machine tool spindle as a 5th 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 630 3D touch probe system

Function

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.

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).

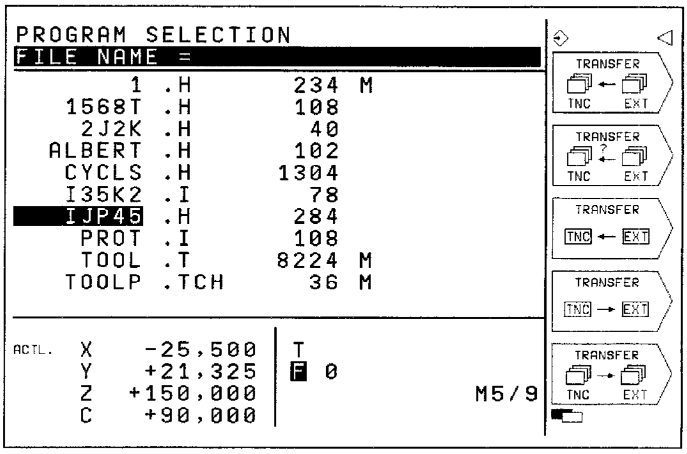
Input range:	0 to 360°
Input resolution:	0.1°

9 External Data Transfer

The TNC features an RS-232-C data interface for transferring data to and from other devices. It can be used in the PROGRAMMING AND EDITING operating mode and in a program run mode.

Possible applications:

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



```
PROGRAM SELECTION
FILE NAME =
         1 .H        234  M
     1568T .H        108
      2J2K .H         40
    ALBERT .H        102
     CYCLS .H       1304
     I35K2 .I         78
     IJP45 .H        284
      PROT .I        108
      TOOL .T       8224  M
     TOOLP .TCH       36  M

ACTL.  X   -25,500 | T
       Y   +21,325 | F 0
       Z  +150,000 |               M5/9
       C   +90,000 |
```

Fig. 9.1: Screen layout for external data transfer

9.1 Functions for External Data Transfer

PGM MGT	Select file management.
EXT	Select functions for external data transfer.

Function	Soft key
Transfer all programs from the storage medium into the TNC	TRANSFER TNC ← EXT
Display programs for transfer into the TNC	TRANSFER TNC ?← EXT
Transfer selected program into the TNC	TRANSFER TNC ← EXT
Transfer selected program to an external device	TRANSFER TNC → EXT
Transfer all programs stored in the memory of the TNC to an external device	TRANSFER TNC → EXT
Display program directory of the files stored in the external device	SHOW EXT DIRECTORY

Aborting data transfer

To abort a data transfer process, press DEL.

If you are transferring data between two TNC controls, the receiving control must be started first. To load programs in HEIDENHAIN dialog format, the program input setting under MOD must be on HEIDENHAIN (see page 10-3).

Blockwise transfer

In the PROGRAM RUN/FULL SEQUENCE and SINGLE BLOCK modes you can run programs that exceed the memory capacity of the TNC by using blockwise transfer with simultaneous execution (see page 3-6).

9.2 Pin Layout and Connecting Cable for Data Interface

RS-232-C/V.24 Interface

HEIDENHAIN devices

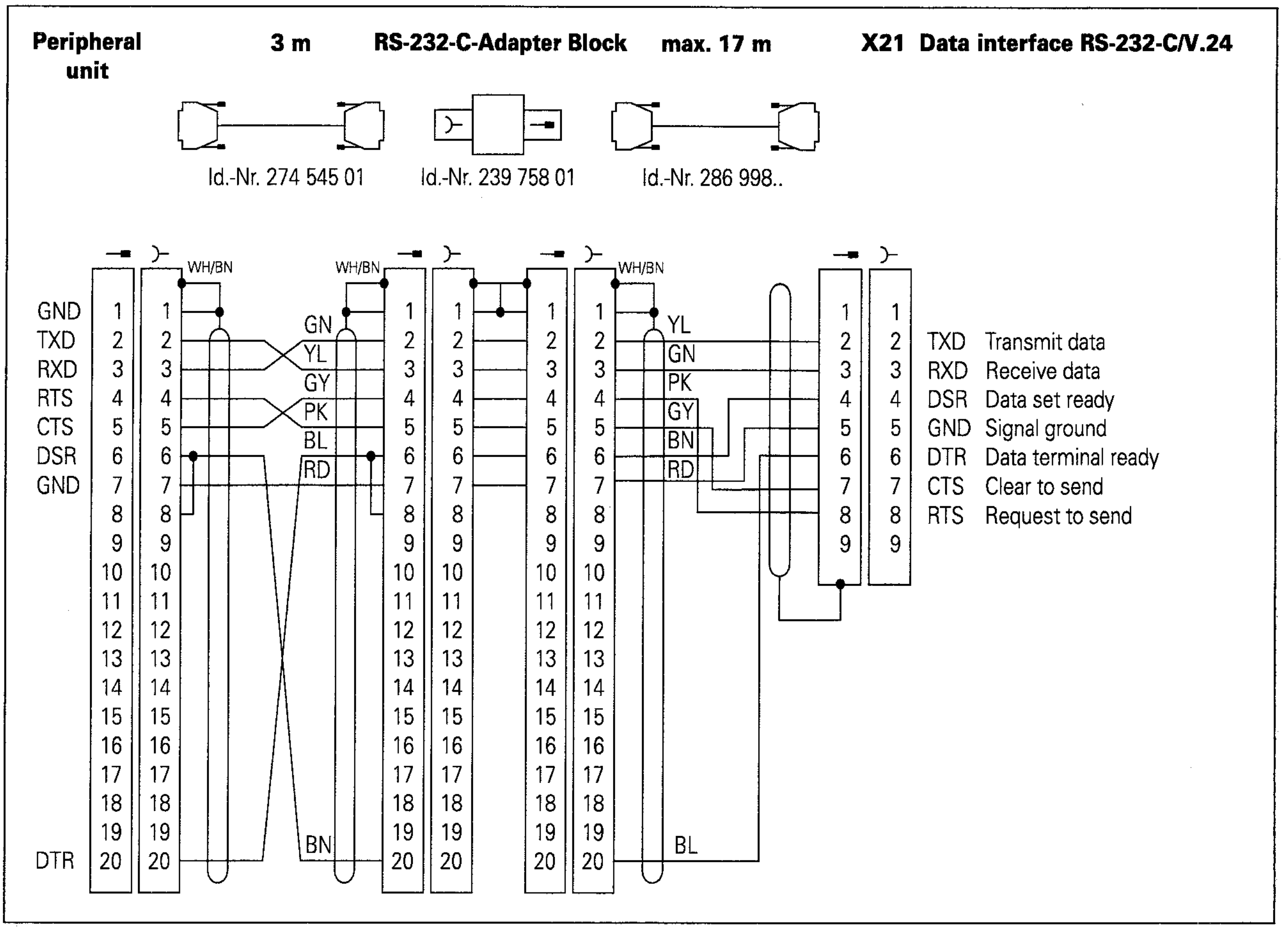


Fig. 9.2: Pin layout of the RS-232-C/V.24 interface for HEIDENHAIN devices

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

Non-HEIDENHAIN devices

The connector pin layout on a non-HEIDENHAIN device may differ considerably from that on a HEIDENHAIN device. The pin layout will depend on the unit and the type of data transfer.

9.3 Preparing the Devices for Data Transfer

HEIDENHAIN devices

HEIDENHAIN devices (FE floppy disk unit and ME magnetic tape unit) are designed for use with 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 the TNC with data transfer cable
- Switch on the FE
- Insert a diskette into the upper drive
- Format the diskette if necessary
- Set the interface (see page 10-5)
- Transfer the data

The baud rate can be selected on the FE 401 floppy disk unit.

Non-HEIDENHAIN devices

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

Adapting a non-HEIDENHAIN device to a TNC

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

Adapting the TNC to a non-HEIDENHAIN device

Set user parameter 5020.

10 MOD Functions, HELP Function

The MOD function provides additional display and input possibilities. The available MOD functions are selected via soft keys, or directly with ENT.

MOD function	Soft key
Machine-specific user parameters	USER PARAM.
Set traverse limits	AXIS LIMITS
Set up the data interface	RS 232 SETUP
NC and PLC software numbers, show free memory	INFO SYSTEM
Enter code number	🔑

10.1 Calling and Exiting the MOD Functions

To select the MOD function

MOD	Press the MOD key.
AXIS LIMITS	If necessary, select the desired MOD function by soft key.

Once you have activated the MOD function you can immediately select

- the position display type
- the programming format
- the unit of measure

To leave the MOD function

END □	Press the END block key.

10.2 Machine-Specific User Parameters

The machine tool builder can assign functions to up to 16 USER PARAMETERS. For more detailed information, refer to the operating manual for the machine tool.

10.3 Selecting the Programming Format and Unit of Measure

The MOD function PROGRAM INPUT lets you choose between programming in HEIDENHAIN plain language format or ISO format:

- To program in HEIDENHAIN format:
  Set the PROGRAM INPUT function to HEIDENHAIN
- To program in ISO format:
  Set the PROGRAM INPUT function to ISO

This MOD function determines whether coordinates are displayed in millimeters or inches.

- Metric system: e.g. X = 15.789 (mm)
  MOD function CHANGE MM/INCH
  The value is displayed with 3 places after the decimal point
- Inch system: e.g. X = 0.6216 (inch)
  MOD function CHANGE MM/INCH
  The value is displayed with 4 places after the decimal point

10.4 Setting the Axis Traverse Limits

The MOD function AXIS LIMITS allows you to set limits to axis traverse within the machine's maximum working envelope.

Possible application:
to protect an indexing fixture from tool collision.

The maximum traverse range is defined by software limit switches. This range can be additionally limited through MOD function AXIS LIMITS. With this function you can enter the maximum traverse positions for the positive and negative axis directions. These values are referenced to the scale datum.

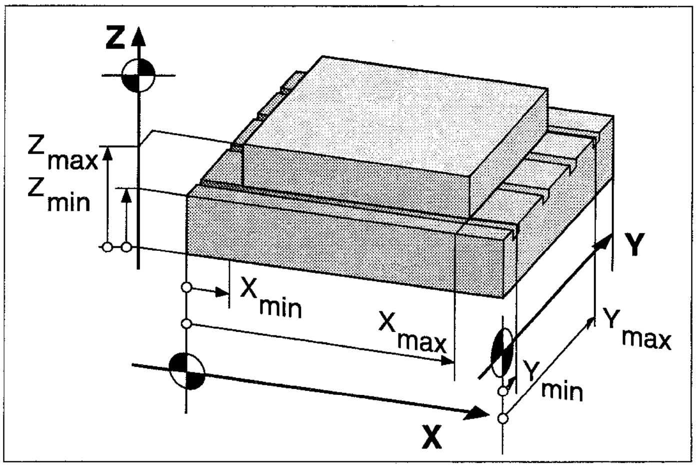


Fig. 10.1: Traverse limits on the workpiece

Working without additional traverse limits

To allow certain coordinate axes to use their full range of traverse, enter the maximum traverse of the TNC 370 (+/- 30 000 mm) as the AXIS LIMIT.

To find and enter the maximum traverse:

Select POSITION DISPLAY REF.

Move the spindle to the desired positive and negative end positions of the X, Y and Z axes.

Write down the values, noting the algebraic sign.

MOD AXIS LIMITS	Select the MOD functions.

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 limits values.
- Traverse range limits and software limit switches become active as soon as the reference marks are crossed over.
- In every axis the TNC checks whether the negative limit is smaller than the positive one.
- The reference positions can also be captured directly with the function "Actual Position Capture" (see page 4-24).

10.5 Setting the External Data Interface

Two functions are available for setting the external data interface:

- BAUD RATE
- RS-232-C INTERFACE

Use the soft key RS-232 SETUP to select the functions.

BAUD RATE

The baud rate is the speed of data transfer in bits per second.

Permissible baud rates (enter with the numerical keys):
110, 150, 300, 600, 1200, 2400, 4800, 9600, 19200, 38400, 57 600, 115 200 baud

The ME 101 has a baud rate of 2400.

RS-232-C Interface

The proper setting depends on the connected device.

Use the ENT key to select the baud rate.

External device	RS-232-C INTERFACE =
HEIDENHAIN FE 401 and FE 401 B floppy disk units	FE
HEIDENHAIN ME 101 magnetic tape unit *(no longer in production)*	ME
Non-HEIDENHAIN units such as printers, tape punchers, PCs without TNC.EXE	EXT
No data transfer, such as for digitizing without transfer of digitized data, or operation without connecting a device	-empty-

10.6 Position Display Types

The positions indicated in Fig. 10.2 are:

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

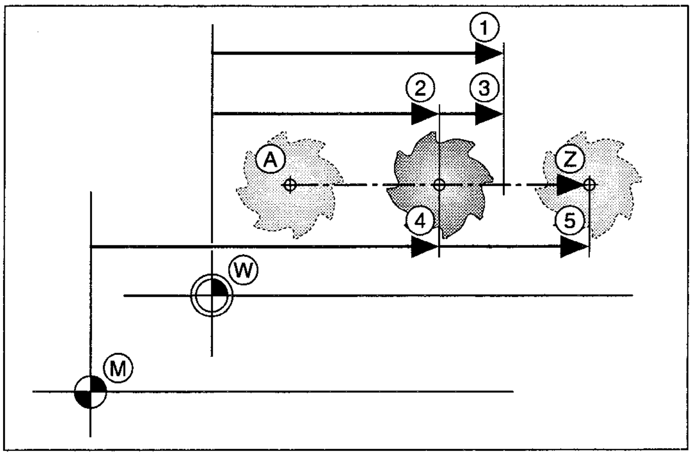


Fig. 10.2: Characteristic positions on the workpiece and scale

As standard, the TNC shows only actual positions. With the additional position display you can also display the following types of position values:

- 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 (difference between the nominal and actual positions) ③ .... LAG
- Reference position (actual position as referenced to the scale datum) ④ .... REF
- Distance remaining to the programmed position (difference between actual and target positions) ⑤ .... DIST.

Select the desired information with the ENT key. It is then displayed directly in the status field.

10.7 Code Numbers

A code number is required for access to certain functions:

Function	Code number
Cancel erase/edit protection (status P)	86357
Select user parameters	123
Timers for: Control ON Program run Spindle ON	857282

Enter code numbers in the dialog field after the corresponding MOD function is selected.

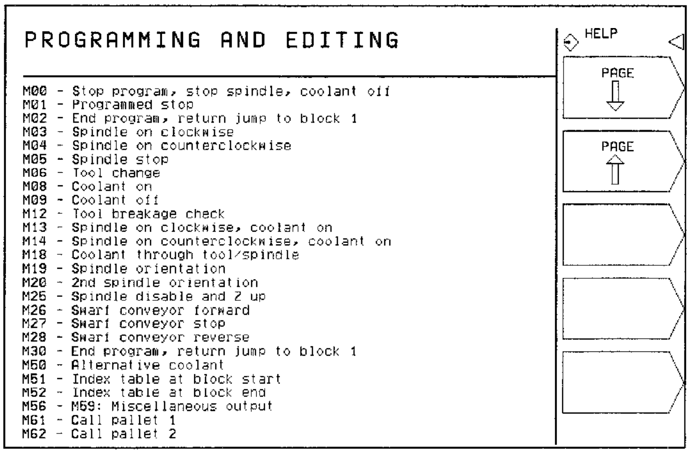
10.8 NC and PLC Software Numbers, Free Memory

Select this MOD function to see the software numbers of the NC and PLC, and the amount of free memory (in number of characters).

10.9 HELP Function

The supplementary operating mode HELP serves as an abridged programming directory and provides an overview of the functions that are available on the TNC. It also displays the soft key structure of each operating mode.

The individual HELP texts comprise several pages and can be selected with the corresponding soft keys. You can use the arrow soft keys to scroll through the pages on the screen.



```
PROGRAMMING AND EDITING

M00 - Stop program, stop spindle, coolant off
M01 - Programmed stop
M02 - End program, return jump to block 1
M03 - Spindle on clockwise
M04 - Spindle on counterclockwise
M05 - Spindle stop
M06 - Tool change
M08 - Coolant on
M09 - Coolant off
M12 - Tool breakage check
M13 - Spindle on clockwise, coolant on
M14 - Spindle on counterclockwise, coolant on
M18 - Coolant through tool/spindle
M19 - Spindle orientation
M20 - 2nd spindle orientation
M25 - Spindle disable and Z up
M26 - Swarf conveyor forward
M27 - Swarf conveyor stop
M28 - Swarf conveyor reverse
M30 - End program, return jump to block 1
M50 - Alternative coolant
M51 - Index table at block start
M52 - Index table at block end
M56 - M59: Miscellaneous output
M61 - Call pallet 1
M62 - Call pallet 2
```

Fig. 10.3: Screen layout with active HELP function

HELP text	Soft key
List of G functions	G
List of D functions	D
List of M functions	M
List of address letters	ADDR LETTER
List of the globally effective Q parameters for machining cycles	Q

Selecting and leaving HELP

To select HELP

HELP	Press the HELP key.
M	Select the desired HELP text with the corresponding soft key.

To leave HELP

HELP	Press the HELP key again.

11 Tables and Overviews

11.1 General User Parameters

General user parameters are machine parameters which affect the behavior of the TNC. These parameters set such things as:

- Dialog language
- Interface behavior
- Traversing speeds
- Machining sequence
- Effect of the overrides

Some machine parameters have more than one function. The input value of such parameters is the sum of the individual values. For these machine parameters the individual values are preceded by a plus sign (***+0***, ***+1***, etc.)

Selecting general user parameters

General user parameters are selected with code number 123 in the MOD functions.

The MOD functions also include machine-specific user parameters.

External data transfer

Control characters for blockwise transfer

MP5010.0	End of program (bits 0 to 7): **Decimal value for ASCII character (0 to 32382)**
	Start of program (bits 8 to 15):
MP5010.1	Data input (bits 0 to 15):
MP5010.2	Data output (bits 0 to 15):
MP5010.3	Start of command block (bits 0 to 7):
	End of command block (bits 8 to 15):
MP5010.4	Positive acknowledgement (bits 0 to 7):
	Negative acknowledgement (bits 8 to 15):
MP5010.5	End of data transfer: (bits 0 to 15):

Adapt TNC interface to external device

MP5020	7 data bits (ASCII code, 8th bit = parity): ***+0***
	8 data bits (ASCII code, 9th bit = parity): ***+1***
	Block Check Character (BCC) any: ***+0***
	Block Check Character (BCC) not allowed: ***+2***
	Transmission stop with RTS active: ***+4***
	Transmission stop with RTS not active: ***+0***
	Transmission stop with DC3 active: ***+8***
	Transmission stop with DC3 not active: ***+0***
	Character parity even: ***+0***
	Character parity odd: ***+16***
	Character parity not desired: ***+0***
	Character parity desired: ***+32***
	$1^{1}/_{2}$ stop bits: ***+0***
	2 stop bits: ***+64***
	1 stop bit: ***+128***
	1 stop bit: ***+192***
	Example:
	To adapt the TNC interface to an external non-HEIDENHAIN device, use the following setting:
	8 data bits, any BCC, transmission stop with DC3,
	even character parity, character parity desired, 2 stop bits:
	1+0+8+0+32+64 = 105 (entry value for MP 5020)

Interface type

MP5030	Standard: ***0***
	Interface for blockwise transfer: ***1***

3D touch probes and digitizing

Signal transmission type

MP6010	Cable transmission: ***0*** Infrared transmission: ***1***

Probing feed rate for triggering touch probes

MP6120	***80 to 3000 [mm/min]***

Maximum distance to first probe point

MP6130	***0 to 99 999.999 [mm]***

Safety clearance over probing point during automatic measurement

MP6140	***0 to 99 999.999 [mm]***

Rapid traverse for probe cycle with triggering touch probes

MP6150	***1 to 300 000 [mm/min]***

TNC displays, TNC editor

Programming station

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

Block number increment for ISO programming

MP7220	***0 to 150***

Dialog language

MP7230	German: ***0*** English: ***1***

Configure tool table

MP7260	Not active: ***0*** Number of tools per tool table: ***1 to 99***

Configure tool pocket table

MP7261	Not active: ***0*** Number of pockets per table: ***1 to 99***

Tool pocket table: Number of reserved pockets next to special tools

MP72640 to 3

Configuration of tool tables; column number in the tool table for (do not execute: 0)

MP7266.0	Tool name – NAME: ***0 to 22***
MP7266.1	Tool length – L: ***0 to 22***
MP7266.2	Tool radius – R: ***0 to 22***
MP7266.3	*not used*
MP7266.4	Tool length oversize – DL: ***0 to 22***
MP7266.5	Tool radius oversize – DR: ***0 to 22***
MP7266.6	*not used*
MP7266.7	Tool locked – TL: ***0 to 22***
MP7266.8	Replacement tool – RT: ***0 to 22***
MP7266.9	Maximum tool life – TIME1: ***0 to 22***
MP7266.10	Maximum tool life for TOOL CALL – TIME2: ***0 to 22***
MP7266.11	Current tool age – CUR. TIME: ***0 to 22***
MP7266.12	Tool description – DOC: ***0 to 22***
MP7266.13	Number of teeth – CUT.: ***0 to 22***
MP7266.14	Wear tolerance for tool length – LTOL: ***0 to 22***
MP7266.15	Wear tolerance for tool radius – RTOL: ***0 to 22***
MP7266.16	Cutting direction – DIRECT.: ***0 to 22***
MP7266.17	PLC status – PLC: ***0 to 22***
MP7266.18	Tool end offset in addition to MP6530 from top of stylus – TT:L-OFFS.: ***0 to 22***
MP7266.19	Tool offset between stylus center and tool center – TT:R-OFFS.: ***0 to 22***
MP7266.20	Breakage tolerance for tool length – LBREAK: ***0 to 22***
MP7266.21	Breakage tolerance for tool radius – RBREAK: ***0 to 22***

TNC displays, TNC editor

Configuration of tool pocket table; column number in the tool table for (do not execute: 0)

MP7267.0	Tool number – T: ***0 to 5***
MP7267.1	Special tool – ST: ***0 to 5***
MP7267.2	Fixed pocket – F: ***0 to 5***
MP7267.3	Pocket locked – L: ***0 to 5***
MP7267.4	PLC status – PLC: ***0 to 5***

Settings for MANUAL OPERATION mode

MP7270	Do not display feed rate: ***+0*** Display feed rate: ***+1*** Spindle speed S and M functions still active after STOP: ***+0*** Spindle speed S and M functions no longer active after STOP: ***+2***

Decimal character

MP7280	The decimal character is a comma: ***0*** The decimal character is a point: ***1***

Display step for coordinate axes

MP7290	0.001 mm: ***0*** 0.005 mm: ***1***

Reset status display, Q parameters and tool data

MP7300	Do not reset Q parameters and status display: ***+0*** Reset Q parameters and status display with M02, M30 and N99999: ***+1*** Do not activate last active tool data after power interruption: ***+0*** Activate last active tool data after power interruption: ***+4***

Graphics display

MP7310	Projection in 3 planes according to ISO 6433, projection method 1: ***+0*** Projection in 3 planes according to ISO 6433, projection method 2: ***+1*** Do not rotate coordinate system for graphics display: ***+0*** Rotate coordinate system for graphics display by 90°: ***+2***

Machining and program run

Position display in the tool axis

MP 7285	Display referenced to tool datum: ***0*** Display referenced to tool face: ***1***

Effect of Cycle G72 SCALING FACTOR

MP7410	SCALING FACTOR effective in 3 axes: ***0*** SCALING FACTOR effective in the working plane only: ***1***

Tool compensation data in the touch probe cycle (G55)

MP7411	Overwrite current tool data with the calibrated data of the 3D touch probe: ***0*** Retain current tool data: ***1***

Behavior of Cycle G57 ROUGH-OUT

MP7420	Mill channel around the contour—clockwise for islands and counterclockwise for pockets: ***+0*** Mill channel around the contour—clockwise for pockets and counterclockwise for islands: ***+1*** First mill contour channel, then rough out: ***+0*** First rough out, then mill contour channel: ***+2*** Combine compensated contours: ***+0*** Combine uncompensated contours: ***+4*** Complete one process for all infeeds before switching to the other process: ***+0*** Mill channel and rough-out for each pecking depth before going to the next depth: ***+8***

Overlap factor for POCKET MILLING cycle (G75/G76) and CIRCULAR POCKET cycle (G77/G78)

MP7430	*0.1 to 1.414*

Effect of M functions

MP7440	Program stop with M06: ***+0*** No program stop with M06: ***+1*** No cycle call with M89: ***+0*** Modal cycle call with M89: ***+2*** Program stop with M functions: ***+0*** No program stop with M functions: ***+4***

Angle of directional change up to which tool will move at constant feed rate (corner with R0, inside corner also radius-compensated)

This parameter is effective both during operation with servo lag as well as with feed forward control.

MP7460	***0.0000 to 179.9999 [°]***

Coordinate display for rotary axes

MP7470	Angle display up to ± 359.999°: ***0*** Angle display up to ± 30.000°: ***1***

Override behavior and electronic handwheels

Override

MP7620	Feed rate override not effective when rapid traverse key depressed: ***+0*** Feed rate override effective when rapid traverse key depressed: ***+1*** Feed rate override not effective when rapid traverse key and machine axis direction button depressed: ***+0*** Feed rate override effective when rapid traverse key and machine axis direction button depressed: ***+4*** 1 % increments for feed rate override: ***+0*** 0.01 % increments for feed rate override: ***+8***

Handwheel type

MP7640	No handwheel: ***0*** HR 330 with additional keys – the 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 12 additional keys: ***4*** Multi-axis handwheel with additional keys: ***5***

11.2 Miscellaneous Functions (M Functions)

Miscellaneous functions with predetermined effect

M	Effect of M function	Effective at block start	end	Page
M00	Program STOP / spindle STOP / coolant OFF		•	3-4
M01	Optional program STOP		•	3-7
M02	Program STOP / spindle STOP / coolant OFF clear status display (depending on machine parameter) / return to block 1		•	3-4
M03	Spindle ON clockwise	•		
M04	Spindle ON counterclockwise	•		
M05	Spindle STOP		•	
M06	Tool change / program STOP (depending on machine parameter) spindle STOP		•	3-4
M08	Coolant ON	•		
M09	Coolant OFF		•	
M13	Spindle ON clockwise/coolant ON	•		
M14	Spindle ON counterclockwise/coolant ON	•		
M30	Same as M02		•	3-4
M89	Vacant miscellaneous function **or**	•		
	cycle call, modally effective (depending on machine parameter)		•	8-3
M90	Smoothing corners	•		5-35
M91	Within the positioning block: Coordinates referenced to machine datum	•		5-38
M92	Within the positioning block: Coordinates referenced to position defined by machine manufacturer, such as tool change position	•		5-38
M93	Within the positioning block: Coordinates referenced to current tool position. Effective in blocks with G40, G43, G44	•		4-15
M94	Limit display of rotary axis to value under 360°	•		5-39
M95	*Reserved*		•	
M96	*Reserved*		•	
M97	Machine small contour steps		•	5-36
M98	Completely machine open contours		•	5-37
M99	Blockwise cycle call		•	8-3
M109	Constant contouring speed at tool cutting edge on circular arcs (increase and decrease feed rate)	•		5-40
M110	Constant contouring speed at tool cutting edge on circular arcs (feed rate decrease only)	•		5-40
M111	Reset M109/M110		•	5-40
M126	Optimized traverse of rotary axes	•		5-39
M127	Reset M126	•		5-39

Vacant miscellaneous functions

Vacant M functions are defined by the machine tool builder. They are described in the operating manual of your machine.

M function	Effective at	
	start of block	end of block
M07	•	
M10		•
M11	•	
M12		•
M15	•	
M16	•	
M17	•	
M18	•	
M19		•
M20	•	
M21	•	
M22	•	
M23	•	
M24	•	
M25	•	
M26	•	
M27	•	
M28	•	
M29	•	
M31	•	
M32		•
M33		•
M34		•
M35		•
M36	•	
M37	•	
M38	•	
M39	•	
M40	•	
M41	•	
M42	•	
M43	•	
M44	•	
M45	•	
M46	•	
M47	•	
M48	•	
M49	•	
M50	•	

M function	Effective at	
	start of block	end of block
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

The 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 the parameters Q100 to Q107 to transfer values from the PLC to an NC program.

Tool radius: Q108

The radius of the current tool 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 the M function last programmed for the spindle.

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: Q113

The value of Q113 specifies whether the highest-level NC program (for nesting with PGM CALL) is programmed in millimeters or inches.
After NC start, Q113 is set as follows:

Unit of measurement in main program	Parameter value
Millimeters	Q113 = 0
Inches	Q113 = 1

Current tool length: Q114

The current value of the tool length is assigned to Q114.

Coordinates from probing during program run

Parameters Q115 to Q118 are assigned the coordinates of the spindle position upon probing during a programmed measurement with the 3D touch probe. The length and radius of the stylus are not included in these coordinates.

Coordinate axis	Parameter
X axis	Q115
Y axis	Q116
Z axis	Q117
Axis IV	Q118

Deviation between actual value and nominal value during automatic tool measurement with the TT 120 touch probe

Actual-nominal deviation	Parameter
X axis	Q115
Y axis	Q116

Current tool radius compensation

The current tool radius compensation is assigned to parameter Q123 as follows:

Current tool radius compensation	Parameter value
G40	Q123 = 0
G41	Q123 = 1
G42	Q123 = 2
G43	Q123 = 3
G44	Q123 = 4

11.4 Features, Specifications and Accessories

Technical information	
Description	Contouring control for up to four axes, with oriented spindle stop
Components	Logic unit, TE 370 or TE 371 keyboard, monochrome flat luminescent screen with soft keys
Data interface	RS-232-C / V.24
Simultaneous axis control for contour elements	Straight lines: up to 4 axes
	Circles: up to 2 axes
	Helices: 3 axes
Parallel programming	For editing one part program while the TNC is running another (not available with FK free contour programming)
Test run	Internally and with test run graphics
Program types	HEIDENHAIN conversational format ISO programs Tool table (program TOOL.T) Pocket table (program TOOLP.TCH)
Program memory	Battery-buffered for up to 64 programs Capacity: approximately 5000 program blocks (depending on the block lengths)
Tool definitions	Up to 254 tools in one program or up to 99 tools in TOOL.T

TNC Specifications	
Block processing time	10 ms per block (three-dimensional straight line without radius compensation)
Control loop cycle time	6 ms
Data transfer rate	Max. 115 200 baud
Ambient temperature	Operation: 0° to 45° C (32° to 113° F) Storage: –30° to 70° C (–22° to 158° F)
Traverse range	Max. ± 100 m (2540 in.)
Traversing speed	Max. 300 m/min (11 811 ipm)
Spindle speed	Max. 99 999 rpm
Input range	As fine as 1 μm (0.0001 in.) or 0.001°

Programmable functions	
Contour elements	Straight line Chamfer Circular arc Circle center Circle radius Tangentially connecting arc Corner rounding
Free contour programming	For all contour elements not dimensioned for NC
Program jumps	Subprograms Program section repetition Main program as subprogram
Fixed cycles	Peck drilling and tapping (also with synchronized spindle) Rectangular and circular pocket milling Slot milling Milling pockets and islands from a list of subcontour elements
Coordinate transformations	Datum shift Mirroring Rotation Scaling factor
3D touch probe system	Touch probe functions for setting datums and for automatic workpiece measurement Digitizing 3D surfaces with a triggering touch probe (optional; only available with conversational programming) Automatic tool measurement with the TT 120 touch probe
Mathematical functions	Basic operations +, −, x, ÷ Trigonometric functions sin, cos, tan and arctan Square roots ($\sqrt{a}$) and Root sum of squares ($\sqrt{a^2 + b^2}$) Logical comparisons greater than, less than, equal to, not equal to

Accessories

FE 401 floppy disk unit	
Description	Portable bench-top unit
Applications	All TNC contouring controls, TNC 131, TNC 135
Data interfaces	Two RS-232-C/V.24 interface ports
Data transfer rate	• TNC: 2400 to 38 400 baud • PRT: 110 to 9600 baud
Floppy disk drives	Two drives, one for copying, capacity 795 kilobytes (approx. 25 000 blocks), up to 256 files
Diskettes	3.5", DS DD, 135 TPI

TS 220 and TS 630 triggering 3D touch probes		
Description		Touch probe system with ruby tip and stylus with rated break point, standard shank for spindle insertion
Models	TS 220:	Cable transmission, integrated interface
	TS 630:	Infrared transmission, separate transmitting and receiving units
Spindle insertion	TS 220:	manual
	TS 630:	automatic
Probing repeatability		Better than 1 µm (0.000 04 in.)
Probing speed		Max. 3 m/min (118 ipm)

TT 120 triggering 3D touch probe	
Description	Touch probe system with hardened, stainless-steel probing element (steel plate), class of protection: IP 67
Interface	Connected to TNC via 5 V supply
Installation	Fixed installation within the machine working space
Probing speed	Max. 3m/min (118 ipm)

Electronic handwheels	
HR 130	Integrable unit
HR 330	Portable version, transmission via cable. Includes axis address keys, rapid traverse key, safety switch, emergency stop button

11.5 TNC Error Messages

The TNC automatically generates error messages when it detects problems such as

- Incorrect data input
- Logical errors in the program
- Contour elements that are impossible to machine
- Incorrect use of the touch probe system

An error message containing a program block number was caused by an error in that block or in the preceding block. To clear a TNC error message, first correct the problem and then press the CE key.

Some of the more frequent TNC error messages are explained in the following list.

TNC error messages during programming

ENTRY VALUE INCORRECT

- Enter a correct label number.
- Press the correct key.

EXT. IN-/OUTPUT NOT READY

The external device is not correctly connected.

FURTHER PROGRAM ENTRY IMPOSSIBLE

Erase some old files to make room for new ones.

JUMP TO LABEL 0 NOT PERMITTED

Do not program L0.0.

LABEL NUMBER ALLOCATED

Label numbers can only be assigned once.

TNC error messages during test run and program run

ANGLE REFERENCE MISSING

- Define the arc and its end points unambiguously.
- If you enter polar coordinates, define the polar coordinate angle correctly.

ARITHMETICAL ERROR

You have attempted to calculate with illegal values.

- Define values within the range limits.
- Choose probe positions for the 3D touch probe that are farther separated.
- Calculations must be mathematically possible.

AXIS DOUBLE PROGRAMMED

Each axis can only have one value for position coordinates.

BLK FORM DEFINITION INCORRECT

- Program the MIN and MAX points according to the instructions.
- Choose a ratio of sides less than 84:1.
- When programming with G30/G31, copy the blank form into the main program with %.

CHAMFER NOT PERMITTED

- A chamfer block must be inserted between two straight line blocks.
- A running program cannot be changed.
- Do not edit a program while it is being transferred or executed.

CIRCLE END POS. INCORRECT

- Enter complete information for tangential arcs.
- Enter end points that lie on the circular path.

CYCL INCOMPLETE

- Define the cycle with all data in the proper sequence.
- Do not call coordinate transformation cycles.
- Define a cycle before calling it.
- Enter a pecking depth other than 0.

EXCESSIVE SUBPROGRAMMING

- Conclude subprograms with G98 L0.
- Program subprogram calls without repetitions.
- Call program section repeats including the repetitions.
- Subprograms cannot call themselves.
- Subprograms cannot be nested in more than 8 levels.
- Programs cannot be nested as subprograms in more than 4 levels.

FEED RATE IS MISSING

Enter the feed rate for the positioning block.

GROSS POSITIONING ERROR

The TNC monitors positions and movements. If the actual position deviates too greatly from the nominal position, this blinking error message is displayed. Press the END key for a few seconds to acknowledge the error message (warm start).

KEY NON-FUNCTIONAL

This message always appears when you press a key that is not needed for the current dialog.

LABEL NUMBER NOT ALLOCATED

You can only call labels numbers that have been assigned.

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.
- Movements in a rotary axis cannot be graphically simulated.
- Enter a tool axis for simulation that is the same as the axis in G30.

PLANE WRONGLY DEFINED

- Do not change the tool axis while a basic rotation is active.
- Define the main axes for circular arcs correctly.
- Define both main axes for I,J.

PROBE SYSTEM NOT READY

- Orient transmitting/receiving window of TS 630 to face receiving unit.
- Check whether the touch probe is ready for operation.

PROGRAM-START UNDEFINED

- Begin the program only with a G0 G40 G90 block.
- Do not resume an interrupted program at a block with a tangential arc or pole transfer.

RADIUS COMPENSATION UNDEFINED

Enter radius compensation in the first subprogram to Cycle G37 CONTOUR GEOM.

ROUNDING OFF NOT DEFINED

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 go to the beginning of the program by entering GOTO 0.

STYLUS ALREADY IN CONTACT

Before probing, pre-position the stylus so that it is not touching the workpiece surface.

TOOL RADIUS TOO LARGE

Enter a tool radius that
- lies within the given limits, and
- permits the contour elements to be calculated and machined.

TOUCH POINT INACCESSIBLE

Pre-position the 3D touch probe to a point nearer the surface.

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.
- Chamfer length must be positive.

WRONG RPM

Program a spindle speed within the permissible range.

11.6 Address Characters (ISO Programming)

G Functions

Group	G	Function	Effective blockwise	Refer to page
Positioning functions	00	Linear interpolation, Cartesian coordinates, at rapid traverse		5-9
	01	Linear interpolation, Cartesian coordinates		5-9
	02	Circular interpolation, Cartesian coordinates, clockwise		5-17
	03	Circular interpolation, Cartesian coordinates, counterclockwise		5-17
	05	Circular interpolation, Cartesian coordinates, no direction of rotation defined		5-17
	06	Circular interpolation, Cartesian coordinates, tangential connection		5-23
	07	Paraxial positioning block	•	5-41
	10	Linear interpolation, polar, at rapid traverse		5-27
	11	Linear interpolation, polar coordinates		5-27
	12	Circular interpolation, polar coordinates, clockwise		5-29
	13	Circular interpolation, polar coordinates, counterclockwise		5-29
	15	Circular interpolation, polar coordinates, no direction of rotation defined		5-29
	16	Circular interpolation, polar coordinates, tangential connection		5-31
Cycles	04	Dwell time	•	8-38
	28	Mirror image		8-33
	36	Oriented spindle stop		8-39
	37	Definition of the pocket contour		8-16
	39	Cycle for program call, cycle call G79	•	8-3
	54	Datum shift in a part program		8-30
	56	Pilot drilling contour pockets (combined with G37)		8-25
	57	Roughing out contour pockets (combined with G37)		8-17
	58	Contour milling, clockwise (combined with G37)		8-26
	59	Contour milling, counterclockwise (combined with G37)		8-26
	72	Scaling factor		8-36
	73	Rotation of the coordinate system		8-35
	74	Slot milling		8-9
	75	Rectangular pocket milling, clockwise		8-11
	76	Rectangular pocket milling, counterclockwise		8-11
	77	Circular pocket milling, clockwise		8-13
	78	Circular pocket milling, counterclockwise		8-13
	83	Pecking		8-4
	84	Tapping with a floating tap holder		8-6
	85	Rigid tapping		8-8
	79	Cycle call	•	8-3
Selecting the working plane	17	Select plane XY, tool axis Z		5-15
	18	Select plane ZX, tool axis Y		5-15
	19	Select plane YZ, tool axis X		5-15
	20	Tool axis IV		5-15
Chamfers, corner rounding, approaching and departing a contour	24	Chamfer with length R	•	5-12
	25	Corner rounding with radius R	•	5-25
	26	Tangential contour approach with radius R	•	5-6
	27	Tangential contour departure with radius R	•	5-6
	29	Define the last programmed position as a pole		5-27
Definition of the workpiece blank	30	Define MIN point of blank form for graphic simulation		4-19
	31	Define MAX point of blank form for graphic simulation		4-19
	38	Stop program run	•	3-4

Group	G	Function	Effective blockwise	Refer to page
Tool radius compensation	40	No tool radius compensation (R0)		4-16
	41	Tool radius compensation: tool moves at left of contour (RL)		4-16
	42	Tool radius compensation: tool moves at right of contour (RR)		4-16
	43	Paraxial compensation: tool path is lengthened (R+)		4-16
	44	Paraxial compensation: tool path is shortened (R–)		4-16
	51	Next tool number (with central tool file)	•	4-13
	55	Touch probe function	•	7-13
Unit of measurement	70	Inches (at beginning of program)		10-3
	71	Millimeters (at beginning of program)		10-3
Definition of positions	90	Absolute workpiece positions		1-19
	91	Incremental workpiece positions		1-19
	98	Assigning a label number	•	6-2
	99	Tool definition	•	4-7

Other address characters

Address character	Function
%	Begin program or call program with G39
A	Rotate about the X axis
B	Rotate about the Y axis
C	Rotate about the Z axis
D	Parameter definition (program parameter Q)
F	Feed rate
F	Dwell time with G04
F	Scaling factor with G72
G	Preparatory function
H	Polar angle in incremental/absolute dimensions
H	Rotation angle with G73
I	X coordinate of circle center or pole
J	Y coordinate of circle center or pole
K	Z coordinate of circle center or pole
L	Assign a label number with G98
L	Go to a label number
L	Tool length with G99
M	Miscellaneous functions
N	Block number
P	Cycle parameters in fixed cycles
P	Parameters in parameter definitions
Q	Program parameter/Cycle parameter Q
R	Polar coordinate radius
R	Circle radius with G02/G03/G05
R	Rounding radius with G25/G26/G27
R	Chamfer side length with G24
R	Tool radius with G99
S	Spindle speed
S	Spindle orientation with G36
T	Tool definition with G99
T	Tool call
U	Linear movement parallel to the X axis
V	Linear movement parallel to the Y axis
W	Linear movement parallel to the Z axis
X	X axis
Y	Y axis
Z	Z axis
*	End of block

Parameter definitions

D	Function	Refer to page
00	Assign	7-5
01	Addition	7-5
02	Subtraction	7-5
03	Multiplication	7-5
04	Division	7-5
05	Square root	7-5
06	Sine	7-7
07	Cosine	7-7
08	Root sum of squares ($c = \sqrt{a^2 + b^2}$)	7-7
09	If equal, jump	7-8
10	If not equal, jump	7-8
11	If greater than, jump	7-8
12	If less than, jump	7-8
13	Angle from $c \cdot \sin \alpha$ and $c \cdot \cos \alpha$	7-7
14	Error number	7-11
15	Print	7-12
19	Assign PLC markers	7-12

Sequence of Program Steps

Milling an outside contour

Programming step			Key/Function	Refer to section
1	**Create or select program**			
	Input: Program name		PGM MGT	4.4
2	**Define workpiece blank for graphic simulation**		G30/G31	4.4
3	**Define tool(s)**		G99	
	Input:	Tool number	T...	
		Tool length	L...	4.2
		Tool radius	R...	
4	**Call tool data**		T...	
	Input:	Tool number		
		Spindle axis	G17	4.2
		Spindle speed	S...	
5	**Tool change**			
	Input:	Feed rate (rapid traverse)	G00	
		Radius compensation	G40	e.g. 5.4
		Coordinates of the tool change position	X... Y... Z...	
		Miscellaneous function (tool change)	M06	
6	**Move to starting position**			
	Input:	Feed rate (rapid traverse)	G00	
		Radius compensation	G40	5.2/5.4
		Coordinates of the starting position	X... Y... Z...	
		Misc. function (spindle on clockwise)	M03	
7	**Move tool to (first) working depth**			
	Input:	Feed rate (rapid traverse)	G00	5.4
		Coordinate of the (first) working depth	Z...	
8	**Move to first contour point**			
	Input:	Linear interpolation	G01	
		Radius compensation for machining	G41/G42	5.2/5.4
		Coordinates of the first contour point	X... Y...	
		Machining feed rate	F...	
	with smooth approach if desired: program G26 after this block			
9	**Machining to last contour point**			5 to 8
	Input:	Enter all necessary values for each contour element		
10	**Move to end position**			
	Input:	Feed rate (rapid traverse)	G00	
		Cancel radius compensation	G40	5.2/5.4
		Coordinates of the end position	X... Y...	
		Misc. function (spindle stop)	M05	
	with smooth departure if desired: program G27 after this block			
11	**Retract tool**			
	Inputs:	Feed rate (rapid traverse)	G00	
		Coordinate above the workpiece	Z...	5.2/5.4
		Misc. function (end of program)	M02	
12	**End of program**			

Miscellaneous Functions (M Functions)

M functions with predetermined effect

M	**Effect of M function**	**Effective at block**		**Page**
		start	**end**	
M00	Program STOP / spindle STOP / coolant OFF		•	3-4
M01	Optional program STOP		•	3-7
M02	Program STOP / spindle STOP / coolant OFF clear status display (depending on machine parameter) / return to block 1		•	3-4
M03	Spindle ON clockwise	•		
M04	Spindle ON counterclockwise	•		
M05	Spindle STOP		•	
M06	Tool change / program STOP (depending on machine parameter) spindle STOP		•	3-4
M08	Coolant ON	•		
M09	Coolant OFF		•	
M13	Spindle ON clockwise/coolant ON	•		
M14	Spindle ON counterclockwise/coolant ON	•		
M30	Same as M02		•	3-4
M89	Vacant miscellaneous function **or**	•		
	cycle call, modally effective (depending on machine parameter)		•	8-3
M90	Smoothing corners	•		5-35
M91	Within the positioning block: Coordinates referenced to machine datum	•		5-38
M92	Within the positioning block: Coordinates referenced to position defined by machine manufacturer, such as tool change position	•		5-38
M93	Within the positioning block: Coordinates referenced to current tool position. Effective in blocks with G40, G43, G44	•		4-15
M94	Limit display of rotary axis to value under 360°	•		5-39
M95	*Reserved*	•		
M96	*Reserved*	•		
M97	Machine small contour steps		•	5-36
M98	Completely machine open contours		•	5-37
M99	Blockwise cycle call		•	8-3
M109	Constant contouring speed at tool cutting edge on circular arcs (increase and decrease feed rate)	•		5-40
M110	Constant contouring speed at tool cutting edge on circular arcs (feed rate decrease only)	•		5-40
M111	Reset M109/M110		•	5-40
M126	Optimized traverse of rotary axes	•		5-39
M127	Reset M126	•		5-39

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
00	Assign
01	Addition
02	Subtraction
03	Multiplication
04	Division
05	Square root
06	Sine
07	Cosine

D	Function
08	Root sum of squares $c = \sqrt{a^2 + b^2}$
09	If equal, go to label number
10	If not equal, go to label number
11	If greater than, go to label number
12	If less than, go to label number
13	Angle from $c \sin \alpha$ and $c \cos \alpha$
14	Error number
15	Print
19	Assignment PLC

Address Characters

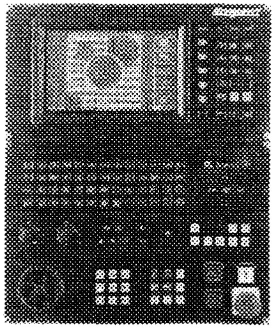
Add.	Function
%	Start of program
%	Program call with G39
A	Rotary motion about X axis
B	Rotary motion about Y axis
C	Rotary motion about Z axis
D	Q parameter definitions
F	Feed rate
F	Dwell time with G04
F	Scaling factor with G72
G	G functions
H	Polar coordinate angle
H	Angle of rotation with G73
I	X coordinate of the circle center/pole
J	Y coordinate of the circle center/pole
K	Z coordinate of the circle center/pole
L	Set a label number with G98
L	Go to a label number
L	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	Polar coordinate radius
R	Circle radius with G02/G03/G05
R	Rounding radius with G25/G26/G27
R	Tool radius with G99
S	Spindle speed
S	Oriented spindle stop with G36
T	Tool definition with G99
T	Tool call
T	Next tool with G51
U	Axis parallel to X axis
V	Axis parallel to Y axis
W	Axis parallel to Z axis
X	X axis
Y	Y axis
Z	Z axis
*****	End of block

Program Example: Milling

Select the program name	PGM MGT
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**

TNC 370



ISO Programming

Operating Modes

The switch-over key sets the screen layout.

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 handwheel 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 that contain all information needed for a positioning move. The blocks are not stored.
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.
Program run/ single block	Each block must be started separately with the machine START button.

Programming:

Programming and editing	This mode allows you to edit HEIDENHAIN conversational and ISO programs, tool tables and pocket tables, and then download or output them over the RS-232-C interface. The interactive graphics provide immediate verification of the blocks as you enter them.
Test run	The **test run** graphics feature allows you to check part programs for errors before doing actual machining.

G Functions

Tool movement

G00	Linear interpolation, Cartesian coordinates, rapid traverse
G01	Linear 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 form 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

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

Coordinate transformations

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
M01	Optional interruption of program run
M02	Stop program run/Spindle stop/Coolant off delete status display (depending on machine parameter) Return to block 1
M03	Spindle ON clockwise
M04	Spindle ON counterclockwise
M05	Spindle stop
M06	Tool change/Spindle stop (depending on machine parameter)/ Stop program run
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, modal
M99	Cycle call, non-modal
M90	Constant contouring speed at inside corners and uncompensated corners
M91	Coordinates in positioning block are referenced to the machine datum
M92	Coordinates in positioning block are referenced to a position defined by the machine builder
M93	*Reserved*
M94	Reduce display of rotary axis to value under 360°
M95	*Reserved*
M96	*Reserved*
M97	Path compensation on outside corners: points of intersection instead of transition arc
M98	End of path compensation, non-modal
M109	Constant contouring speed at tool cutting edge on inside and outside corners
M110	Constant contouring speed at tool cutting edge on inside corners
M111	Feed rate refers to center of tool path (standard behavior)
M126	Optimized traverse of rotary axes
M127	Reset M126

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH
Dr.-Johannes-Heidenhain-Straße 5
D-83301 Traunreut, Deutschland
☎ (08669) 31-0 · Tx 56832
FAX (08669) 5061

289 137 21 · SW07 · 3 · 11/97 · S · Printed in Germany · Subject to change without notice