HEIDENHAIN

TNC 406
TNC 416

NC Software
280 620-xx
280 621-xx
286 180-xx

User's Manual
Conversational
Programming

English (en)
4/2001

Controls on the visual display unit

- Split screen layout
- Switch between machining or programming modes
- Soft keys for selecting functions in screen
- Switching the soft-key rows
- Changing the screen settings (only BC 120)

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
- INCREMENTAL JOG
- POSITIONING WITH MDI
- PROGRAM RUN, SINGLE BLOCK
- PROGRAM RUN, FULL SEQUENCE

Programming modes

- PROGRAMMING AND EDITING
- TEST RUN

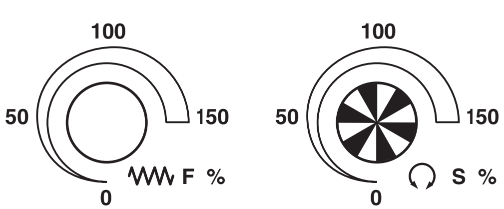
Program/file management, TNC functions

- PGM MGT / PGM NAME — Select programs and files
- CL PGM — Delete programs and files (only TNC 406)
- EXT — Activate external data transfer (only TNC 406)
- MOD — Pocket calculator

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

- Move highlight
- GOTO — Go directly to blocks, cycles and parameter functions

Override control knobs for feed rate/C axis



Programming path movements

- L — Straight line
- CC — Circle center/pole for polar coordinates
- C — Circle with center
- CR — Circle with radius
- CT — Circular arc with tangential connection
- RND — Corner rounding

Electrode data

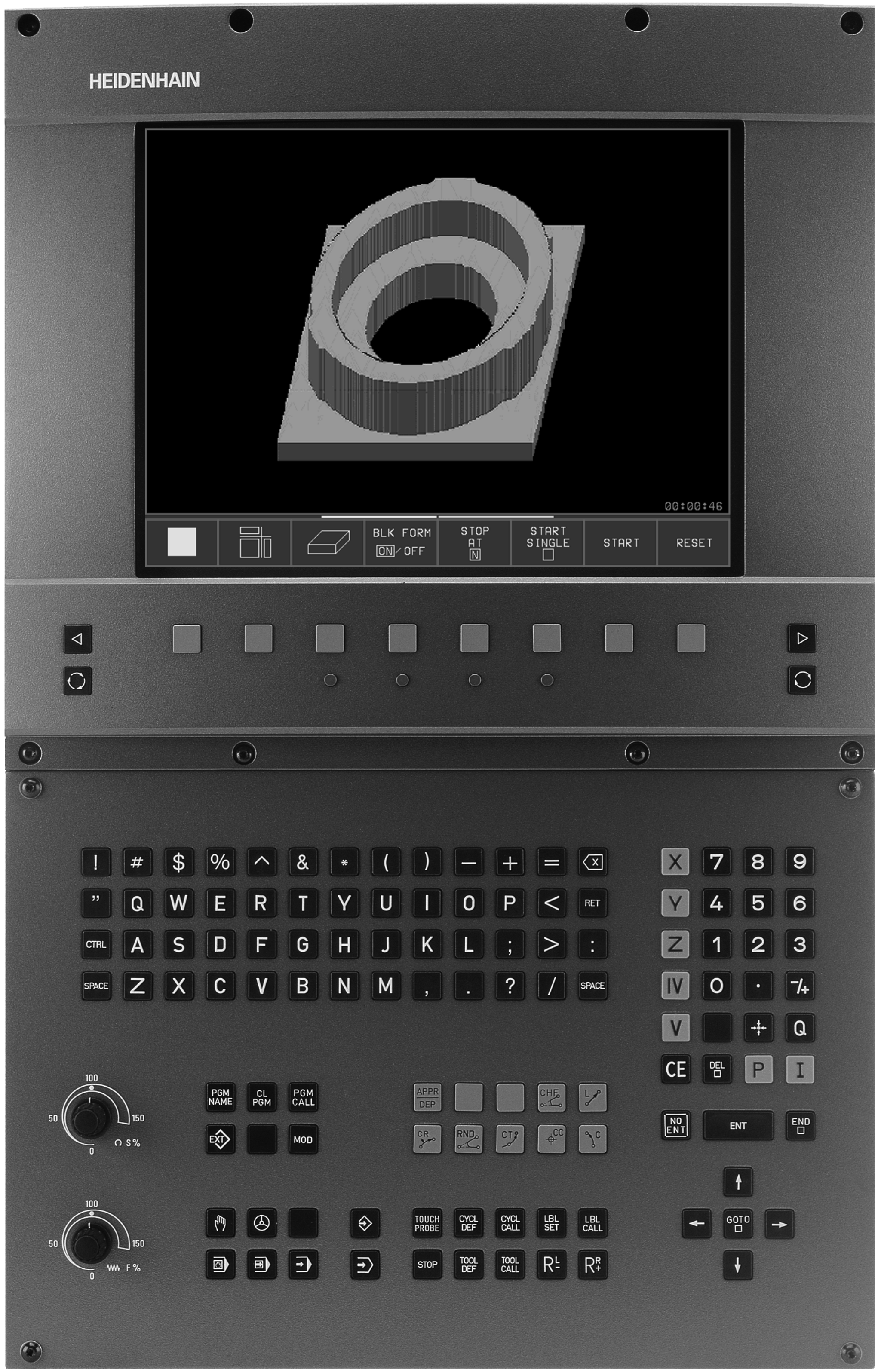
- TOOL DEF / TOOL CALL — Enter and call electrode length and radius
- R+ / R– — Activate electrode radius compensation

Cycles, subprograms and program section repeats

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

Coordinate axes and numbers:Entering and editing

- X . . . V — Select coordinate axes or enter them into the program
- 0 . . . 9 — Numbers
- • — Decimal point
- –/+ — Change arithmetic sign
- P — Polar coordinates
- I — Incremental dimensions
- Q — Q parameters
- Capture actual position
- NO ENT — Skip dialog questions, delete words
- ENT — Confirm entry and resume dialog
- END — End block
- CE — Clear numerical entry or TNC error message
- DEL — Abort dialog, delete program section


HEIDENHAIN
00:00:46
BLK FORM ON/OFF
STOP AT N
START SINGLE
START
RESET
PGM NAME
CL PGM
PGM CALL
MOD
APPR DEP
TOUCH PROBE
CYCL DEF
CYCL CALL
LBL SET
LBL CALL
STOP
TOOL DEF
TOOL CALL
CE
DEL
NO ENT
ENT
END
GOTO
CTRL
SPACE
RET

TNC Models, Software and Features

This manual describes functions and features provided by the TNCs as of the following NC software numbers.

TNC Model	NC Software No.
TNC 406	280 620-12 280 621-12 280 622-12
TNC 416	286 180-06

Location of use

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

New features of the NC software 280 62x-xx and 280 180-xx

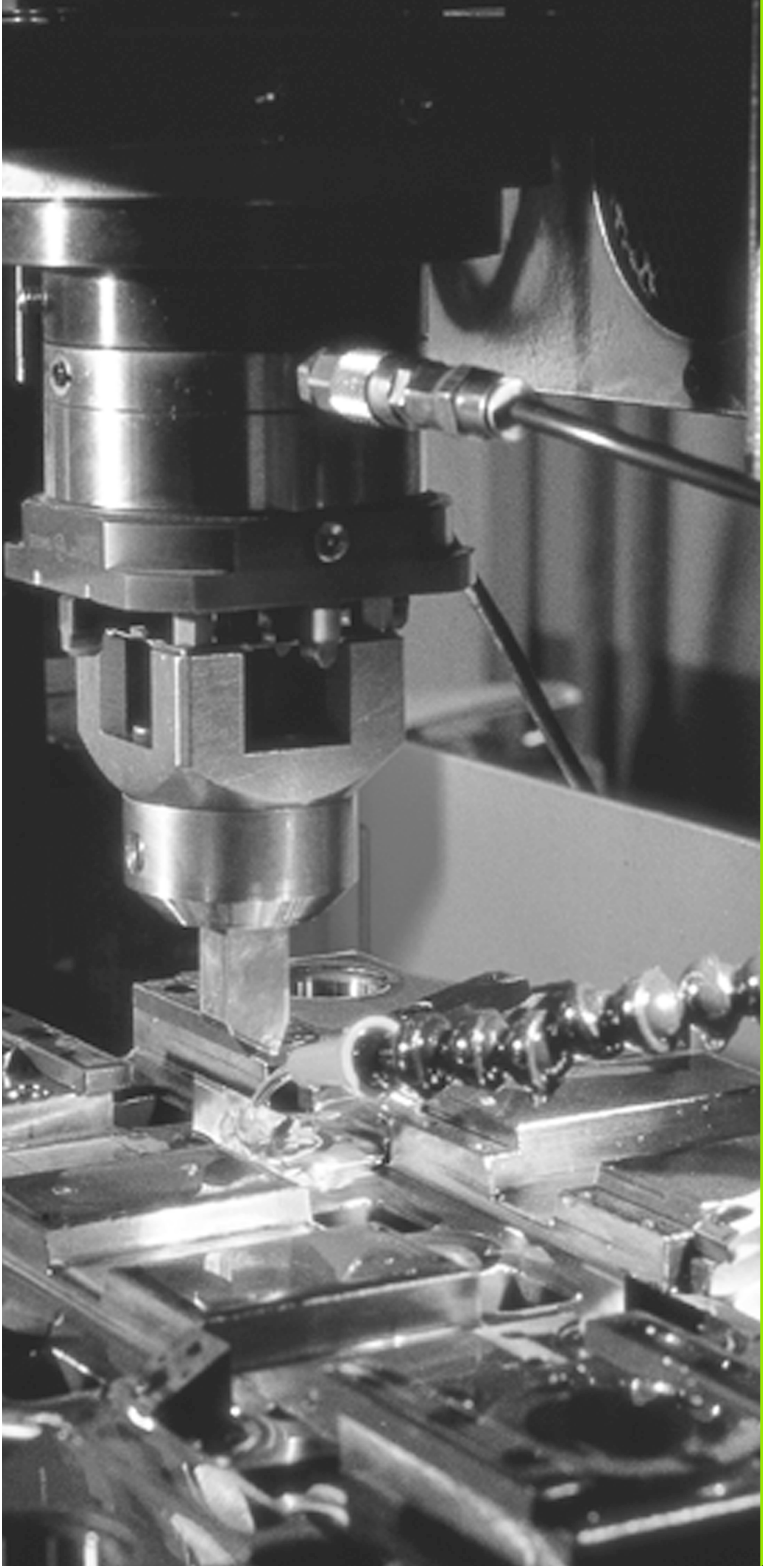
- Cycle 14 CONTOUR GEOMETRY (see also ”Cycle 14 CONTOUR GEOMETRY” on page 137)
- Q parameters for the roughness (see also ”Data from the erosion table” on page 203)
- Q parameters for the gap size (see also ”Gap size LS max when machining which Cycle 1 GENERATOR: Q164” on page 206)
- After manual traverse, the incremental coordinates always refer to the actual position (see also ”Resuming program run with the GOTO key” on page 226)
- Expansion of the tool table with tool pocket number, tool undersize and radius (see also ”Entering electrode data in tables” on page 74)
- Probed values can be written to a datum table as well as to a tool table (see also ”Writing probed values to tables” on page 28)
- Enhancement of functions FN14 and FN15 (see also ”Output of Q Parameters and Messages” on page 197)
- M108/M109 (see Overview of Miscellaneous Functions on the inside rear cover of this manual)

Contents

2 Manual Operation, Setup and Probing Functions 15

3 Positioning with Manual Data Input (MDI) 37

12 MOD Functions 229



1

Introduction

1.1 The TNC 406, the TNC 416

Controls

The TNC 406 and the TNC 416 are shop-floor programmable contouring controls for EDM machines with up to five axes.

Visual display unit and keyboard

The 14-inch color monitor (TNC 406) and 15-inch color monitor (TNC 416) display all information necessary for effective use of the TNC's capabilities.

Program entry is supported by soft keys on the monitor.

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

Programming

The user programs the TNC 406/TNC 416 right at the machine with interactive conversational-type guidance.

Graphics

Workpiece machining can be graphically simulated. Various display modes are available.

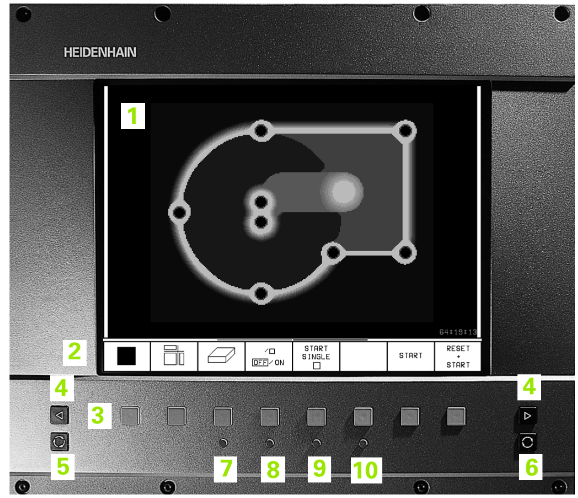
Compatibility

The TNC 406/TNC 416 can execute all programs whose commands belong to the command set of the TNC 406/TNC 416.

1.2 Visual Display Unit and Keyboard

Visual display unit

The TNC 406 is delivered with the BC 110 color monitor (CRT); the TNC 416 can be delivered with the BC 120 color monitor (CRT) or the BF 120 flat-screen color monitor (TFT). The figure at top right shows the keys and controls on the BC 120, and the figure at bottom right shows those of the BF 120.



1 Header

   When the TNC is on, the selected operating modes are shown in the screen header.

2 Soft keys

   In the footer the TNC indicates additional functions in a soft-key row. You can select these functions by pressing the keys immediately below them. The lines immediately above the soft-key row indicate the number of soft-key rows that can be called with the black arrow keys to the right and left. The line representing the active soft-key row is highlighted.

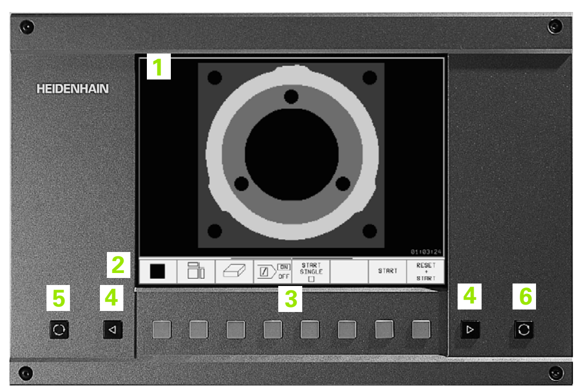
3 Soft key selector keys
4 Switching the soft-key rows
5 Setting the screen layout
6 Shift key for switchover between machining and programming modes

Keys on BC 120 only

7 Screen demagnetization; Exit main menu for screen settings
8 Select main menu for screen settings:
   - In the main menu: Move highlight downward
   - In the submenu: Reduce value or move picture to the left or downward

9
   - In the main menu: Move highlight upward
   - In the submenu: Increase value or move picture to the right or upward

10
   - In the main menu: Select submenu
   - In the submenu: Exit submenu



Main menu dialog	Function
BRIGHTNESS	Adjust brightness
CONTRAST	Adjust contrast
H-POSITION	Adjust horizontal position
V-POSITION	Adjust vertical position
V-SIZE	Adjust picture height

Main menu dialog	Function
SIDE-PIN	Correct barrel-shaped distortion
TRAPEZOID	Correct trapezoidal distortion
ROTATION	Correct tilting
COLOR TEMP	Adjust color temperature
R-GAIN	Adjust strength of red color
B-GAIN	Adjust strength of blue color
RECALL	No function

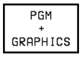
The BC 110 and BC 120 are sensitive to magnetic and electromagnetic noise, which can distort the position and geometry of the picture. Alternating fields can cause the picture to shift periodically or to become distorted.

Screen layout

You select the screen layout yourself: In the TEST RUN mode of operation, for example, you can have the TNC show program blocks in the left window while the right window displays programming graphics. You could also display the tool status in the right window instead, or display only program blocks in one large window. The available screen windows depend on the selected operating mode.

To change the screen layout:

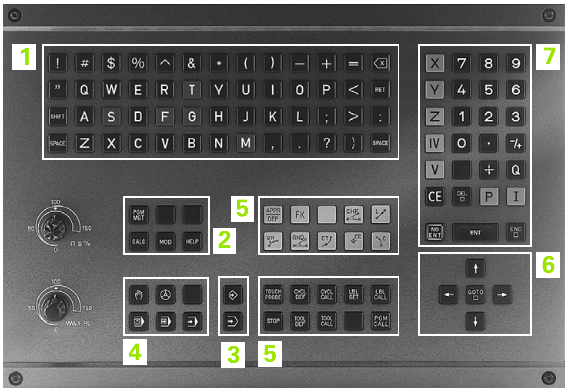
Press the SPLIT SCREEN key: The soft-key row shows the available layout options (see "Modes of Operation" on page 6).



Select the desired screen layout.

Keyboard

The figure at right shows the keys of the keyboard grouped according to their functions:



1 Alphabetic keyboard for entering text and file names
2 ■ File management
■ MOD functions
3 Programming modes
4 Machine operating modes
5 Initiation of programming dialog
6 Arrow keys and GOTO jump command
7 Numerical input and axis selection

The functions of the individual keys are described on the inside front cover. Machine panel buttons, e.g. NC START, are described in the manual for your machine tool.

1.3 Modes of Operation

Manual Operation, Incremental Jog, and Positioning with Manual Data Input

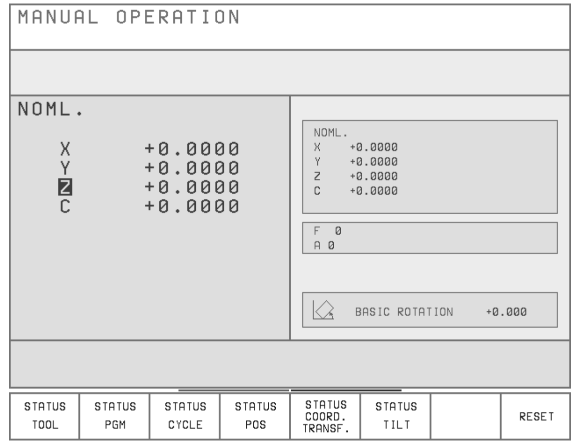
The Manual Operation mode is required for setting up the machine tool. In this mode of operation, you can position the machine axes manually or by increments, set the datums, and tilt the working plane.

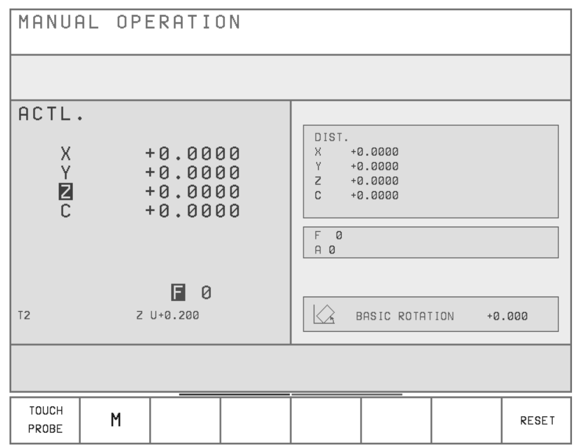
The Incremental Jog mode of operation allows you to move the machine axes manually with the HR electronic handwheel.

Simple traverse movements can be programmed in the Positioning with Manual Data Input (MDI) mode of operation.

Soft keys for selecting the screen layout (see "Screen layout" on page 4)

Screen windows	Soft key
Positions	PGM + POSITION
Left: positions. Right: status display.	PGM POSITION STATUS



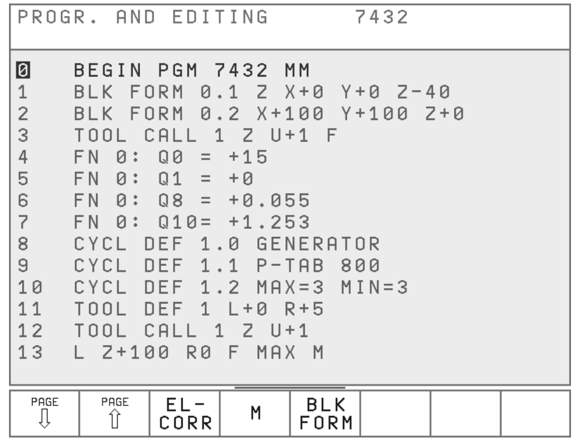


Programming and Editing

In this mode of operation you can write your part programs. The various cycles and Q parameter functions help you with programming and add necessary information.

Soft keys for selecting the screen layout

Screen windows	Soft key
Top: program. Bottom: positions	PGM
Top left: program. Top right: status Bottom: positions	PGM + POSITION

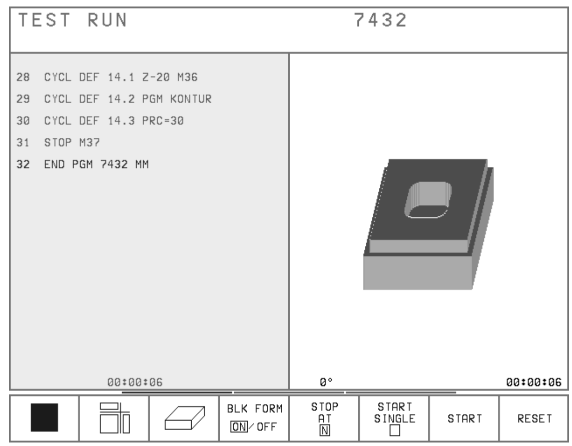


Test Run

In the Test Run mode of operation, the TNC checks programs and program sections for errors, such as geometrical incompatibilities, or missing or incorrect data within the program. This simulation is supported graphically in different display modes.

Soft keys for selecting the screen layout

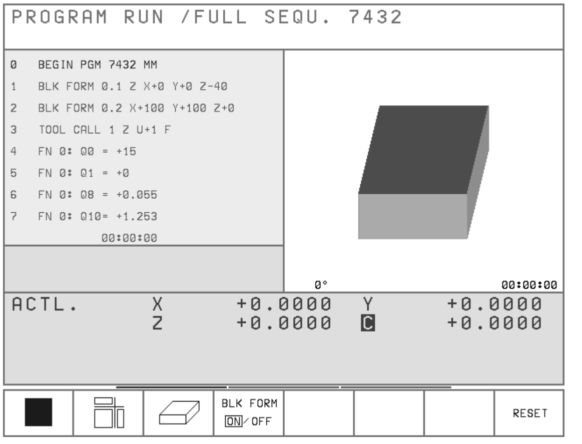
Screen windows	Soft key
Top: program. Bottom: positions	PGM + POSITION
Top left: program. Top right: status Bottom: positions	PGM POSITION STATUS
Left: program. Right: status	PGM + STATUS
Left: program. Right: graphics	PGM + GRAPHICS
Top left: program. Top right: graphics Bottom: positions	PGM POSITION GRAPHICS
Graphics	GRAPHICS



Program Run, Full Sequence and Program Run, Single Block

In the Program Run, Full Sequence mode of operation the TNC executes a part program continuously to its end or to a manual or programmed stop. You can resume program run after an interruption.

In the Program Run, Single Block mode of operation you execute each block separately by pressing the machine START button.



Soft keys for selecting the screen layout

Screen windows	Soft key
Top: program. Bottom: positions	PGM + POSITION
Top left: program. Top right: status Bottom: positions	PGM POSITION STATUS
Top left: program. Top right: graphics Bottom: positions	PGM POSITION GRAPHICS

1.4 Status Display

General status display

Besides the coordinates, the status display also contains the following information:

- Type of position display (ACTL, NOML, etc.)
- Axis is locked (■ on the axis)
- Number of the current electrode T
- Electrode axis
- Feed rate F
- Active miscellaneous functions M
- TNC is in operation (indicated by ✻)
- Name of the selected erosion table
- Permissible power stages (GENERATOR cycle)
- Current power stage

Additional status displays

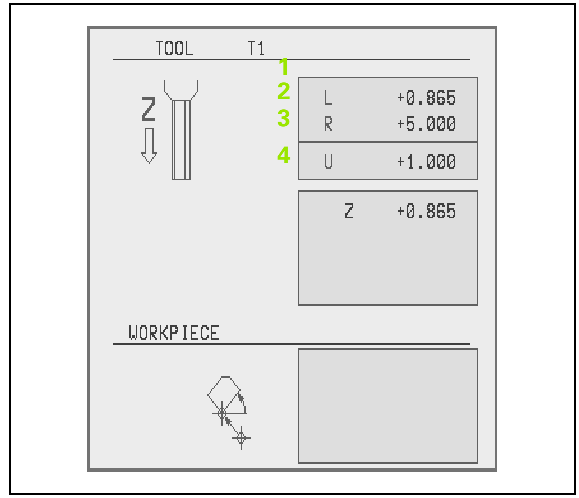
In all modes of operation (except PROGRAMMING AND EDITING), you can split the screen layout to display additional status information in the right screen window:

Additional status display	Soft keys
Information on the current electrode	STATUS TOOL
General program information	STATUS PGM
Information on the current OEM cycle	STATUS CYCLE
Positions and coordinates	STATUS POS.
Active coordinate transformations	STATUS COORD. TRANSF.
Tilting the working plane	STATUS CYCLE

STATUS TOOL

Information on the current electrode

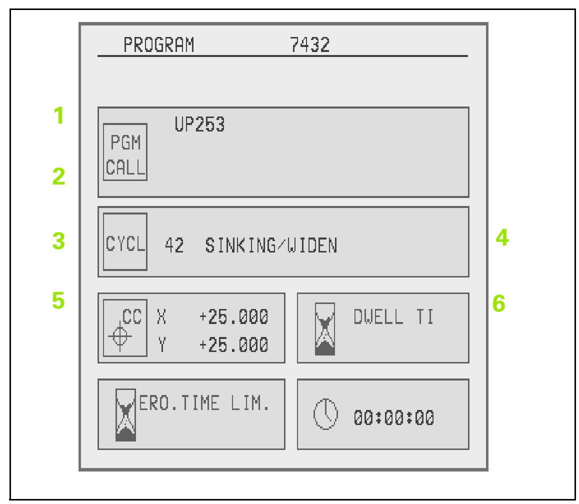
1 Electrode length
2 Electrode radius
3 Electrode undersize
4 Electrode axis



STATUS PGM

General program information

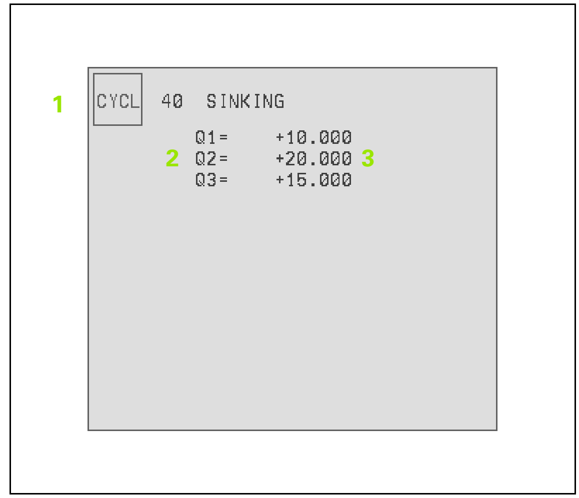
1 Programs called with PGM CALL
2 Active cycle
3 Active circle center
4 Dwell time counter
5 Status for eroding with time limit
6 Operating time



STATUS CYCLE

Information on the current OEM cycle

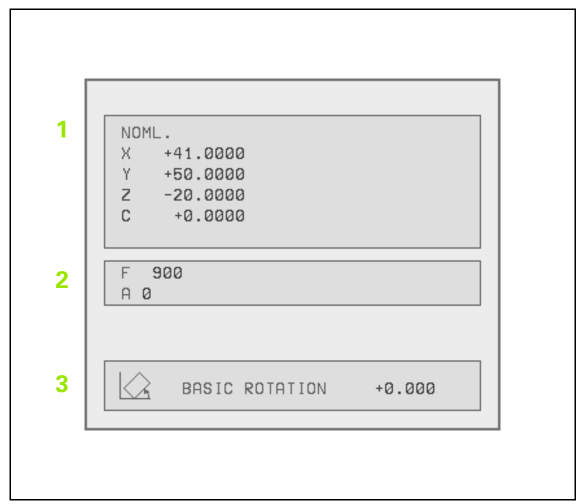
1 Active OEM cycle (number and name)
2 Number of the transfer parameters
3 Content of each transfer parameter



STATUS POS.

Positions and coordinates

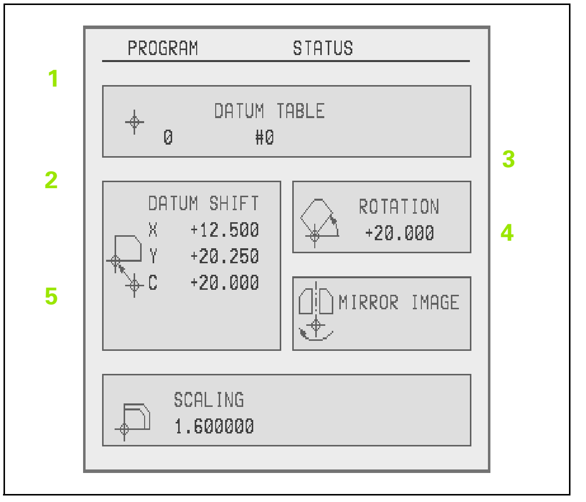
1 Second position display
2 Feed rate and angular position for Cycle 17 DISK
3 Active basic rotation



STATUS COORD. TRANSF.

Active coordinate transformations

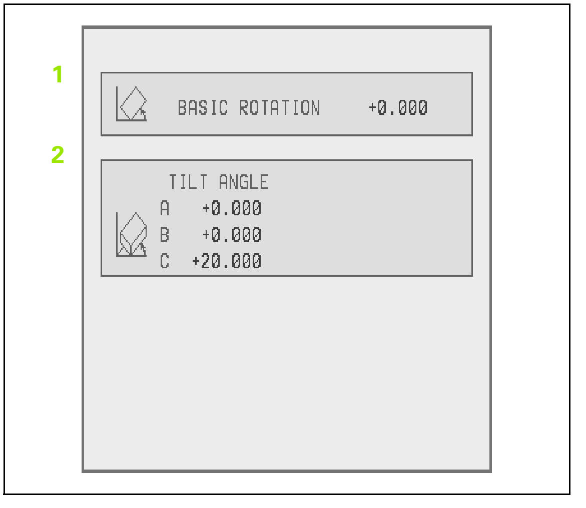
1 Active datum table and active datum number
2 Datum shift
3 Rotation
4 Mirror image
5 Scaling factor



STATUS TILT

Tilting the working plane

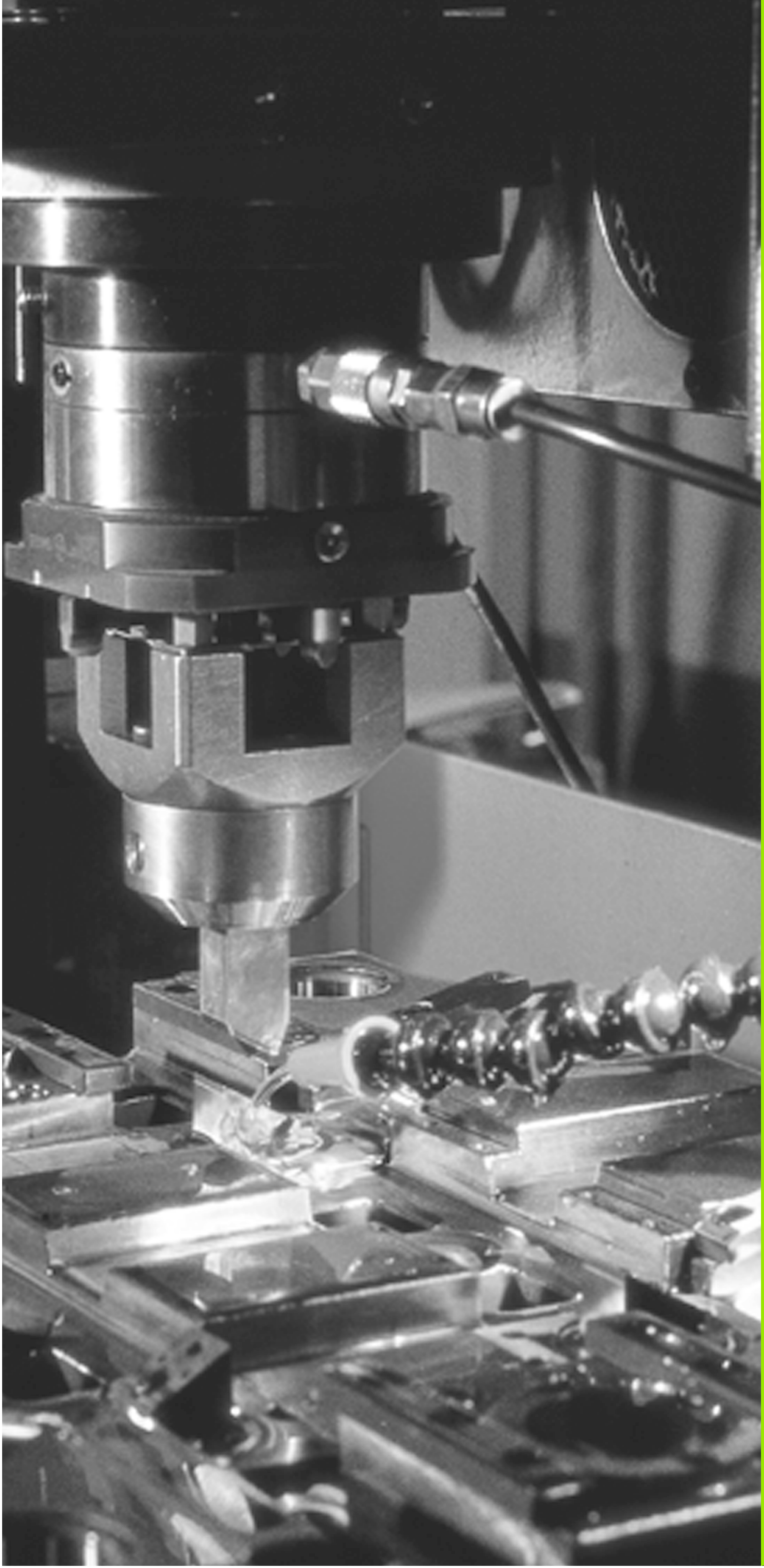
1 Active basic rotation
2 Active tilting angle



1.5 Accessory: Electronic Handwheels from HEIDENHAIN

HR electronic handwheels

The electronic handwheels facilitate precise manual control of the axis slides.
Similar to a conventional machine tool, you move the machine slide a defined distance by turning the handwheel.
A wide range of traverses per revolution is available.
Portable handwheels, such as the HR 410, are connected via cable to the TNC.
Integral handwheels, such as the HR 130, are built into the machine control panel.
Your machine manufacturer can tell you more about the handwheel configuration of your machine.



2

Manual Operation, Setup and Probing Functions

2.1 Switch-on

Switch-on

Switch-on and traversing the reference points can vary depending on the individual machine tool. Refer to your machine manual.

Switch on the power supply for control and machine. The TNC automatically initiates the following dialog

MEMORY TEST

The TNC memory is automatically checked.

POWER INTERRUPTED

CE — TNC message that the power was interrupted — clear the message.

TRANSLATE PLC PROGRAM

The PLC program of the TNC is automatically compiled.

RELAY EXT. DC VOLTAGE MISSING

I — Switch on external dc voltage. The TNC checks the functioning of the EMERGENCY STOP circuit.

MANUAL OPERATION
TRAVERSE REFERENCE POINTS

I — Cross the reference points manually in the displayed sequence: For each axis press the machine START button, or

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

The TNC is now ready for operation in the Manual Operation mode.

> The reference points need only be traversed if the machine axes are to be moved. If you intend only to write, edit or test programs, you can select the Programming and Editing or Test Run modes of operation immediately after switching on the control voltage.
>
> You can traverse the reference marks later by choosing the Manual mode of operation.

2.2 Moving the Machine Axes

Note

The TNC shows the position of up to five machine axes.

The machine manufacturer can enable the position of the fifth axis, for example with the machine axis-direction buttons, with jog increments, with the electronic handwheel or through "PLC positioning".

Contact your machine manufacturer if you need to position a fifth axis.

To traverse with the machine axis direction buttons:

Select the Manual Operation mode.

X

Press the machine axis-direction button and hold it as long as you wish the axis to move, or

X

and

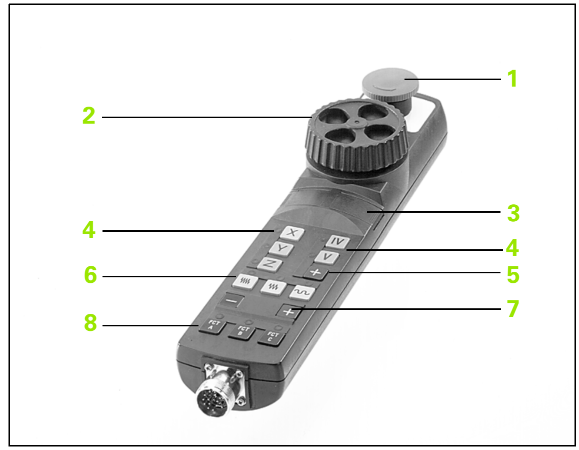
I

Move the axis continuously: Press and hold the machine axis direction button, then press the machine START button

0

To stop the axis, press the machine STOP button.

Traversing with the HR 410 electronic handwheel



The portable HR 410 handwheel is equipped with two permissive buttons. The permissive buttons are located below the star grip.

You can only move the machine axes when an permissive button is depressed (machine-dependent function).

The HR 410 handwheel features the following operating elements:

1 EMERGENCY STOP
2 Handwheel
3 Permissive buttons
4 Axis address keys
5 Actual-position-capture key
6 Keys for defining the feed rate (slow, medium, fast; the feed rates are set by the machine tool builder)
7 Direction in which the TNC moves the selected axis
8 Machine function (set by the machine tool builder)

The red indicators show the axis and feed rate you have selected.

It is also possible to move the machine axes with the handwheel during a program run.

To move an axis:

Select the Jog Increment mode.

Press and hold the permissive button.

Select the axis.

Select the feed rate.

or

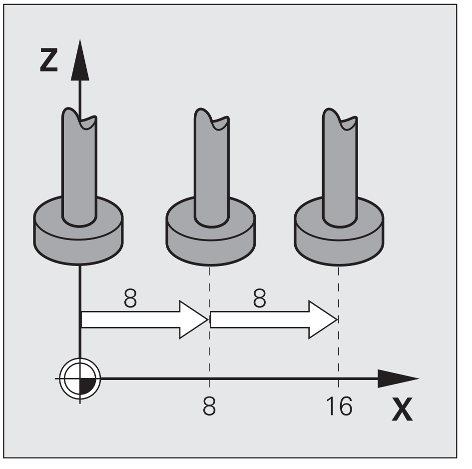
Move the active axis in the positive or negative direction.

If short-circuit monitoring is active: When the electrode makes sparking contact, the TNC stops positioning in the direction of the workpiece, and only permits retracting in the opposite direction. Also, the axes cannot be switched. After they have been retracted at least 10 µm, the TNC switches back to normal Handwheel operation mode. This function is not active while the reference marks are being traversed.

The axes can also be positioned with the electronic handwheel in the PROGRAMMING AND EDITING mode. You must set machine parameter MP7655=1.

Incremental jog positioning

With incremental jog positioning you can move a machine axis by a preset distance.



Incremental Jog Positioning must be enabled by the machine tool builder. Refer to your machine manual.

Select the Jog Increment mode.

INTERPOLATION FACTOR =

4 ENT Enter interpolation factor, i.e. 4

Go to JOG INCREMENT.

JOG INCREMENT =

8 ENT Enter the jog increment in millimeters, i.e. 8 mm.

X The axis moves by the jog increment every time an external axis-direction button is pressed.

Positioning with manual data input (MDI)

Positioning with manual input of the target coordinates is described in Chapter 3 (see "Positioning with Manual Data Input (MDI)" on page 38).

Eroding a workpiece manually

The MANUAL and JOG INCREMENT modes of operation enable you to erode a workpiece manually. This function is especially useful for initial erosion and datum setting. The present gap must be taken into account when setting the datum.

Prerequisite
Cycle 1 GENERATOR must be active.

Procedure

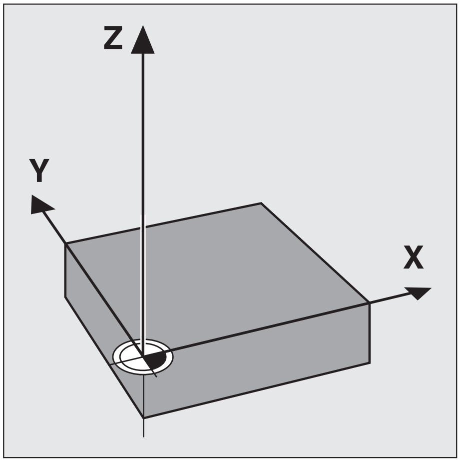
- Select the MANUAL or JOG INCREMENT mode of operation.
- Switch on the generator with M36
- Use the axis direction buttons to preposition the electrode in the working plane. During free run of the electrode, the manual feed rate is effective.
- Move the electrode with the axis direction button until it touches the workpiece. Gap control becomes effective upon contact. The TNC deduces the eroding direction from the axis direction button that was last pressed.

In the MANUAL mode of operation, you can erode up to the limit switch. In the JOG INCREMENT mode of operation, the workpiece is eroded the preset distance.

During erosion you can only move the electrode in the other axes by using the handwheel.

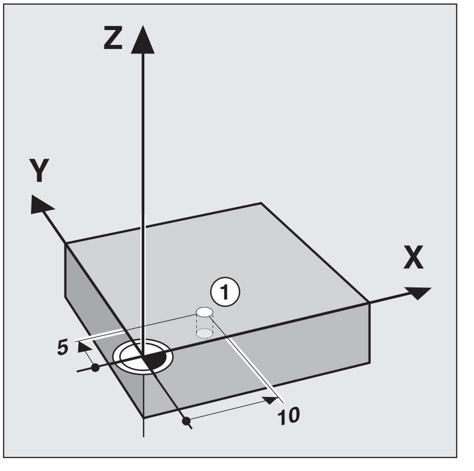
- To end the erosion process, press the machine axis-direction button for the opposite direction.

2.3 Datum Setting



The production drawing identifies a certain form element of the workpiece (usually a corner) as the absolute datum, and usually one or more form elements as relative datums (see "Setting the datum" on page 47). Through the datum setting process, the origin of the absolute or relative coordinate systems is set to these datums: The workpiece – aligned to the machine axes – is brought into a certain position relative to the electrode, and the display is set to zero or the appropriate position value (i.e., to account for the electrode radius).

Example



Coordinates of Point 1:

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

The datum of the rectangular coordinate system is located negative 10 mm on the X axis and negative 5 mm on the Y axis from Point 1.

The fastest, easiest and most accurate way of setting the datum is by using the probing functions for datum determination.

2.4 Calibration and Setup

Using an electrode

An electrode and the probing functions of the TNC 406 can significantly reduce setup time. The TNC 406 offers the following probing functions:

- Compensation of workpiece misalignment (Basic rotation)
- Datum setting
- Measuring
  - lengths and positions on the workpiece
  - angles
  - circle radii
  - circle centers
- Measurements during program run

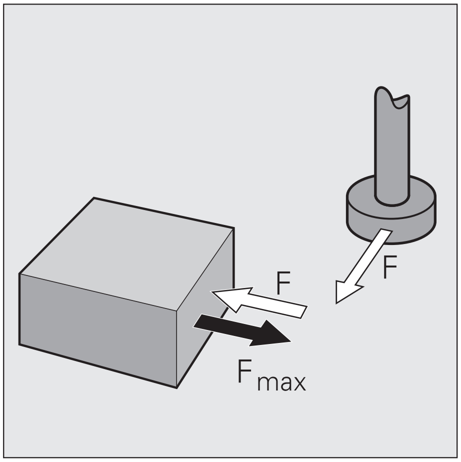
> The TNC must be prepared by the machine tool builder before the probing functions can be used.

In probing functions, the electrode starts moving after the external START button is pressed. The machine tool builder determines the feed rate F for movement towards the workpiece.

When the probing electrode touches the workpiece,

- the TNC stores the coordinates of the probed position,
- the probing electrode stops moving,
- the probing electrode returns to its starting position in rapid traverse.

> Machine parameter 6100 determines whether each probing process is to be executed once or several times (maximum number of probing processes: 5). If you wish to probe several times, the TNC calculates the average of all touch points. This average value is the probing result. (See also "Selecting the General User Parameters" on page 246)



Select the touch probe function

Overview

The following probing functions are available in the Manual and Jog Increment modes:

Function	Soft key
Measuring a basic rotation using a line	PROBING ROT
Manual probing	TOUCH PROBE
Set the datum in any axis	PROBING POS
Set the datum at a workpiece center	PROBING CENTER
Set the datum at a circle center	PROBING CC
Set the datum at a corner	PROBING P
Select the calibration function for the electrode length (second soft-key row)	CAL L
Select the calibration function for the electrode radius (second soft-key row)	CAL R

Select the touch probe function

- Select the Manual Operation or Jog Increment mode.

TOUCH PROBE

- Select the probing function by pressing the TOUCH PROBE soft key. The TNC displays additional soft keys- see table above.

PROBING ROT

- To select the probe cycle: press the appropriate soft key, for example PROBING ROT, and the TNC displays the associated menu.

Calibrating the probing electrode

The probing electrode is to be calibrated in the following situations:

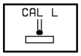
- During commissioning
- When the electrode is changed
- When the probing feed rate is changed
- In case of irregularities, such as those arising when the machine heats up

During calibration, the TNC finds the effective length and the effective radius of the electrode.

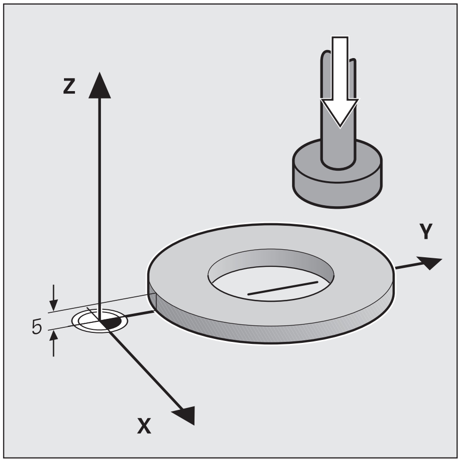
To calibrate the electrode, clamp a ring gauge of known height and inside diameter to the machine table.

To calibrate the effective length:

- Set the datum in the spindle axis such that for the machine tool table Z=0.



- Select the calibration function for the electrode length (second soft-key row).
- Enter the tool axis (with the axis key).
- Datum: Enter the height of the ring gauge.
- Move the probing electrode to a position just above the ring gauge.
- If necessary, change the direction with the cursor keys.
- The electrode probes the surface of the ring gauge: Press the START button.

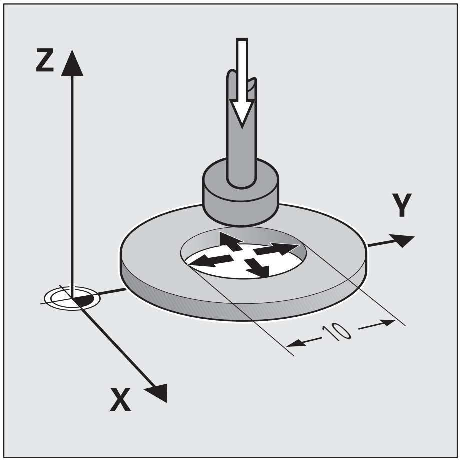


To calibrate the effective radius:

- Position the probing electrode in the hole of the ring gauge.

TOUCH PROBE

- Select the calibration function for the electrode radius (second soft-key row).
- Select the tool axis and enter the radius of the ring gauge.
- To probe the workpiece, press the machine START button four times. The probing electrode touches the hole in each axis direction.
- If you want to terminate the calibration function at this point, press the END soft key.



Displaying calibration values

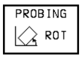
The effective length and radius of the probing electrode are stored in the TNC's memory, and are taken into account when the electrode is used later.
The stored values are displayed on the screen whenever the calibration functions are selected.

Compensating workpiece misalignment

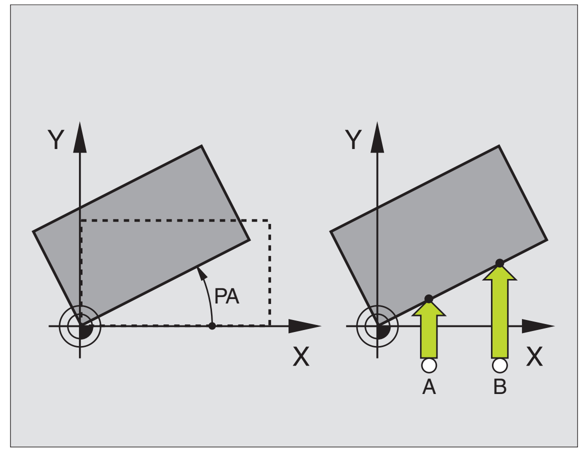
The TNC electronically compensates workpiece misalignment by computing a "basic rotation".

For this purpose, the rotation angle is set to the desired angle with respect to the reference axis in the working plane. If the tilt working plane function is used, the TNC also takes the basic rotation into account in the tilted system.

Measuring the basic rotation



- Select probing function BASIC ROTATION.
- Set ROTATION ANGLE to the nominal value.
- Move the electrode to position **A** near the first probe point **1**.
- Select the probe direction perpendicular to the angle reference axis: Select the axis by soft key.
- To probe the workpiece, press the machine START button.
- Move the electrode to position **B** near the second probe point **2**.
- To probe the workpiece, press the machine START button.



A basic rotation is stored in nonvolatile memory and is effective for all subsequent program runs and graphic simulations.

Displaying a basic rotation

The angle of the basic rotation is shown after ROTATION ANGLE. The rotation angle is also shown in the additional status display window whenever a basic rotation is active.

To cancel a basic rotation:

- Select BASIC ROTATION again.
- Enter a rotation angle of zero and confirm with the ENT key.
- To terminate the probe function, press the END key.

2.5 Datum Setting with a Probing Electrode

Functions for setting the datum

Function	Soft key
Set the datum in any axis	PROBING POS
Manual probing	PROBING DEPTH
Set the datum at a workpiece center	PROBING CENTER
Set the datum at a circle center	PROBING CC
Set the datum at a corner	PROBING P

After probing you can set a new datum or transfer the captured values to a datum or tool table.

Writing probed values to tables

In order to write probed values to datum tables, the tables must be active on your TNC (bit 2 in machine parameter 7224 = 0).

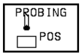
The TNC writes the probed value to a table after the TRANSFER TO TABLE soft key is pressed. You can choose a datum table (NAME.D) as well as a tool table (NAME.T):

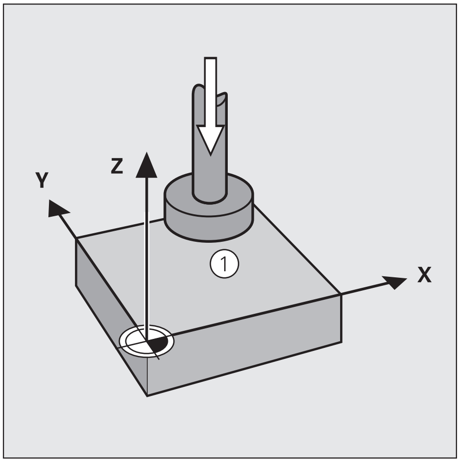
- Select manual probing by pressing the TOUCH PROBE soft key.
- Enter the name of the datum or tool table.
- Enter the datum number or tool number.
- Select the probing function and begin probing.
- Press the TRANSFER TO TABLE soft key for the TNC to write the probed value to the selected table.

Writing probed values to a table while a program is running

You can also write probed values to the TOOL table during program run. Use miscellaneous function M109 to transfer the contents of the Q parameters Q81 to Q84 into the table TOOL.T.
You can also use M108 to read the tool compensation values from the TOOL table into parameters Q81 to Q84 (see also "Q parameters for the datum table: Q81 to Q84" on page 206).

Datum setting in any axis





- Select the probing function by pressing the PROBING POS soft key.
- Move the touch probe to a starting position near the touch point.
- Select the probe axis and direction in which you wish to set the datum, such as Z in direction Z–. Selection is made via soft keys.
- To probe the workpiece, press the machine START button.
- Datum: Enter the nominal coordinate and confirm your entry with ENT.

Manual probing

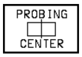
The PROBING DEPTH function enables you to probe the workpiece as often as desired in one axis. At the same time, you can move all remaining axes with the electronic handwheel. This probing function is particularly convenient for finding peaks and valleys.

In this process, the TNC always stores the last point of electrode contact with the workpiece. You can end the probing process with the CYCLE STOP button.

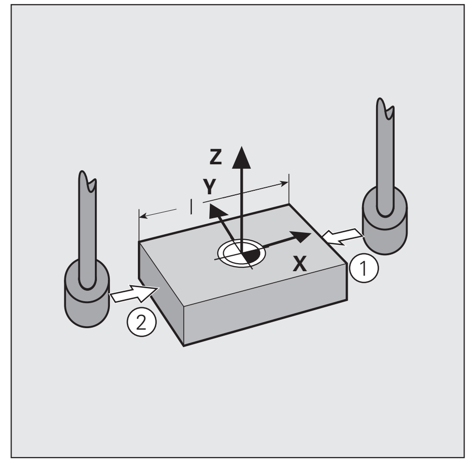
- Select the probing function PROBING DEPTH.
- Move the probing electrode to a starting position near the touch point.
- Set the axis traverse limit, i.e. the maximum permissible traverse of the electrode in the probing axis, and confirm with ENT.
- Select the probe axis and direction in which you wish to set the datum, such as Z in direction Z–.
- Start the probing process. The TNC moves the electrode in the selected axis direction until it makes contact with the workpiece. This coordinate is stored in the TNC memory.
  The probing process is repeated until you end the probing function with CYCLE STOP.
- Use the electronic handwheel to move the electrode in any of the remaining axes to be scanned for peaks or valleys.
- Enter the nominal coordinate of the datum and confirm with ENT.

Workpiece center as datum

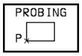
With the function PROBING CENTER, you can find the center of square or rectangular workpieces and set the datum at that point. The workpiece must be aligned paraxially to use this function.



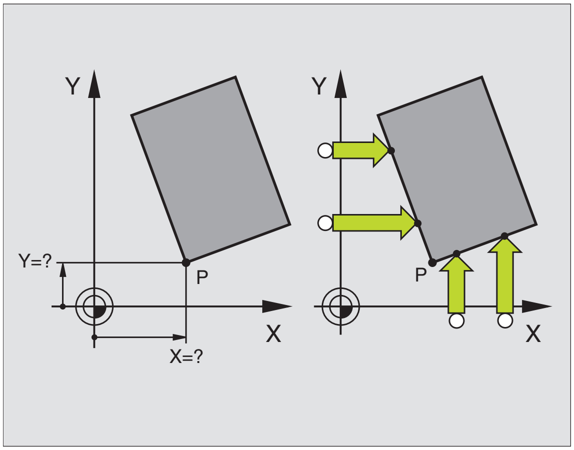
- Select the probing function by pressing the PROBING CENTER soft key.
- Move the probing electrode to a position near the first touch point.
- Select the probing direction via soft key, e.g. X+.
- To probe the workpiece, press the machine START button.
- Move the probing electrode to a position near the second touch point.
- To probe the workpiece, press the machine START button.
- Enter the first coordinate of the datum, for example on the X axis.
- Repeat the process for the third and fourth touch points on the second axis, for example on the Y axis.
- Enter the second coordinate of the datum, for example on the Y axis.
- End the probing function.



Corner as datum

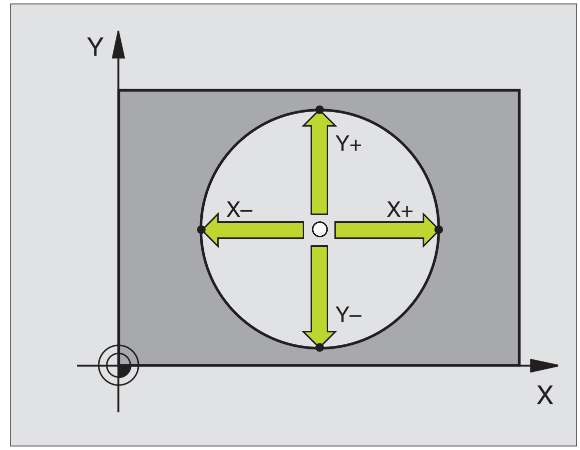


- To select the probe function, press PROBING P.
- Move the probing electrode to a position near the first touch point.
- Select the probing direction via soft key, e.g. X+.
- To probe the workpiece, press the machine START button.
- Position the probing electrode near the second touch point on the same side.
- To probe the workpiece, press the machine START button.
- Probe two points on the next edge in the same manner.
- Datum: Enter both datum coordinates into the menu window, and confirm your entry with the ENT key.
- To terminate the probe function, press the END key.



Circle center as datum

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

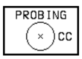


Inside circle

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

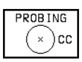
For incomplete circles (circular arcs) you can choose the appropriate probing direction.

- Move the electrode to a position approximately in the center of the circle.

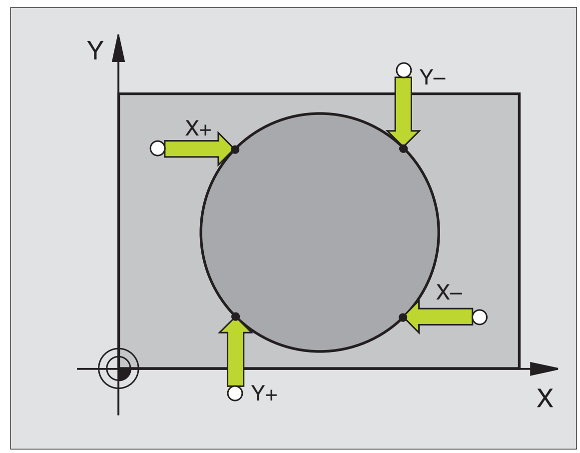


- To select the probe function, press PROBING CC.
- To probe the workpiece, press the machine START button four times. The touch probe touches four points on the inside of the circle.
- Datum: Enter both circle center coordinates into the menu window, and confirm your entry with ENT.
- To terminate the probe function, press the END key.

Outside circle



- To select the probe function, press PROBING CC.
- Move the probing electrode to a position near the first touch point outside of the circle.
- Select the probe direction with a soft key.
- To probe the workpiece, press the machine START button.
- Repeat the probing process for the remaining three points. See figure at lower right.
- Enter the coordinates of the circle center.



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

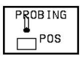
2.6 Measuring with a Probing Electrode

Introduction

An electrode can be used to determine

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

To find the coordinate of a position on an aligned workpiece



- Select the probing function by pressing PROBING POS.
- Move the probing electrode to a starting position near the touch point.
- Select the probe direction and axis of the coordinate. Use the corresponding soft keys for selection.
- To probe the workpiece, press the machine START button.

The TNC shows the coordinates of the touch point as datum.

Finding the coordinates of a corner in the working plane

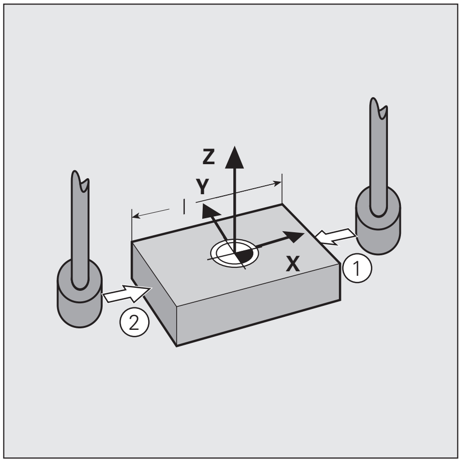
Find the coordinates of the corner point as described under "Corner as datum".

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

Measuring workpiece dimensions

PROBING POS

- Select the probing function by pressing PROBING POS.
- Move the probing electrode to a position near the first touch point **1**.
- Select the probing direction with a soft key.
- To probe the workpiece, press the machine START button.
- If you will need the current datum later, write down the value that appears in the Datum display.
- Set the datum to 0.
- To terminate the dialog, press the END key.
- Select the touch probe function again: Press PROBING POS.
- Move the probing electrode to a position near the second touch point **2**.
- Select the probe direction with the soft keys: Same axis but from the opposite direction.
- To probe the workpiece, press the machine START button.



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 probing function by pressing PROBING POS.
- Probe the first touch point again.
- Set the DATUM to the value that you wrote down previously.
- To terminate the dialog, press the END key.

Measuring angles

You can also use the probing electrode to measure angles in the working plane. You can measure

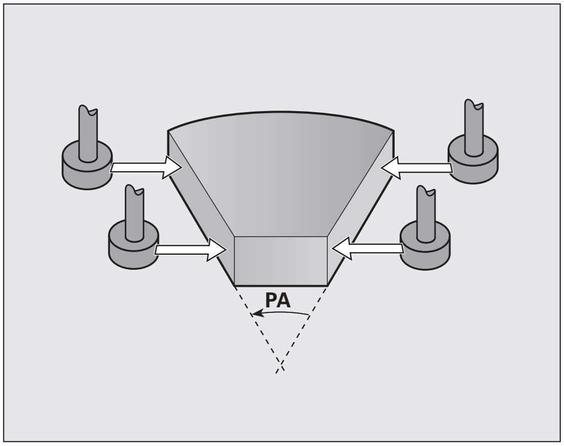
- the angle between the angle reference axis and a workpiece side, or
- the angle between two sides.

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

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

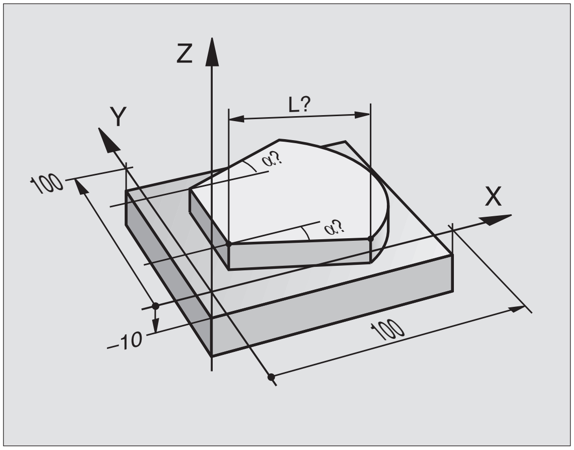
PROBING ROT

- Select the probing function by pressing the PROBING ROT soft key.
- Rotation angle: If you will need the current basic rotation later, write down the value that appears under Rotation angle.
- Make a basic rotation with the side of the workpiece (see "Compensating workpiece misalignment" on page 27).
- Press the PROBING ROT soft key to display the angle between the angle reference axis and the edge of the workpiece as the rotation angle.
- Cancel the basic rotation, or restore the previous basic rotation by setting the Rotation angle to the value that you wrote down previously.



To measure the angle between two workpiece sides:

- Select the probing function by pressing the PROBING ROT soft key.
- Rotation angle: If you will need the current basic rotation later, write down the value that appears under Rotation angle.
- Make a basic rotation with the side of the workpiece (see "Compensating workpiece misalignment" on page 27).
- Probe the second side as for a basic rotation, but do not set the Rotation angle to zero!
- Press the PROBING ROT soft key to display the angle PA between the two sides as the Rotation angle.
- Cancel the basic rotation, or restore the previous basic rotation by setting the Rotation angle to the value that you wrote down previously.



2.7 Entering and Starting Miscellaneous Functions M

Entering values

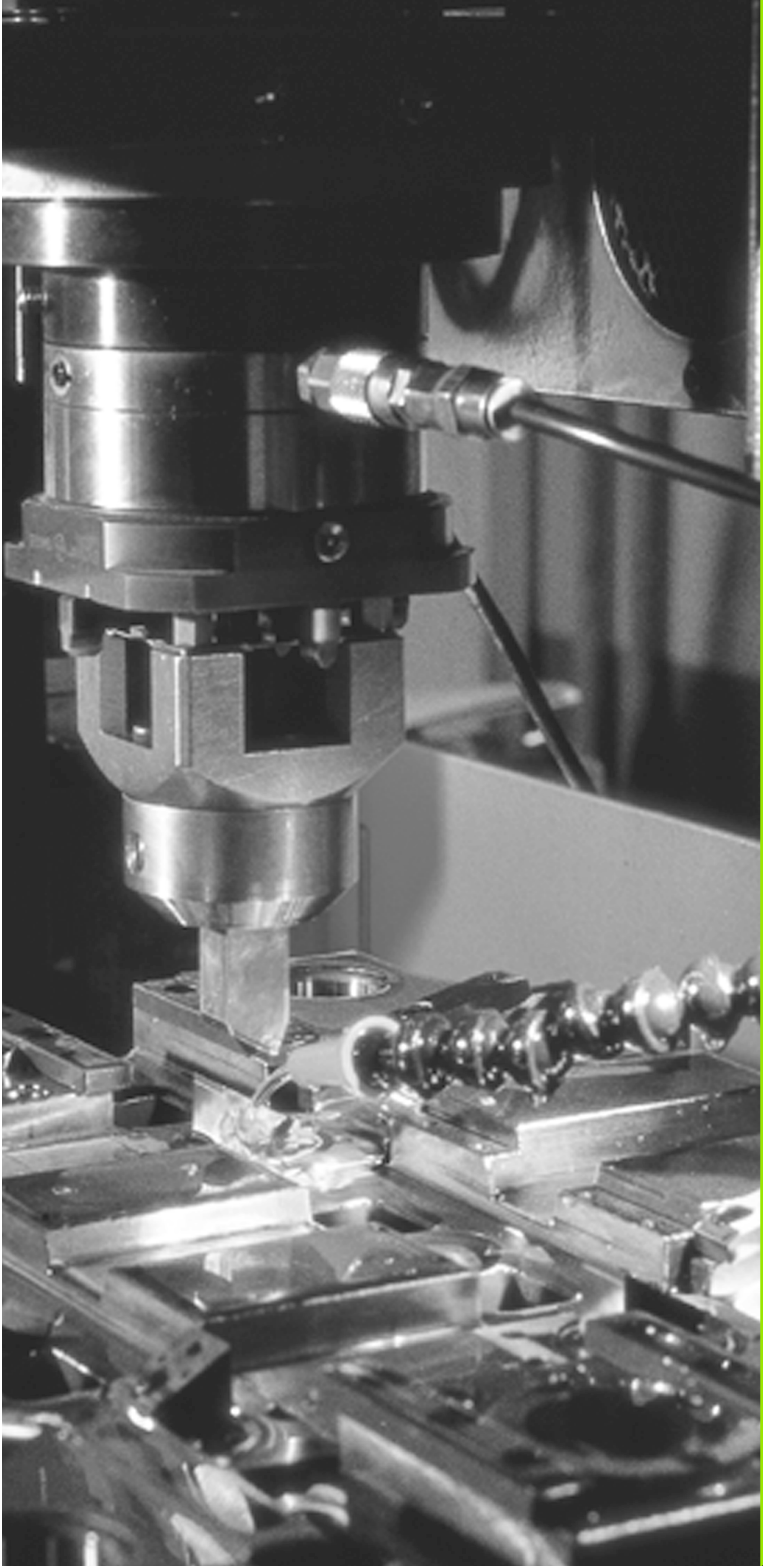
Miscellaneous function M

M	To enter the miscellaneous function, press the M soft key.

MISCELLANEOUS FUNCTION M =

6 ENT	Enter a miscellaneous function, e.g. M6.
I	Start the miscellaneous function.

The machine tool builder determines which miscellaneous functions M are available on your TNC and what function they have. Refer to your machine manual.



3

Positioning with Manual Data Input (MDI)

3.1 Positioning with Manual Data Input (MDI)

The POSITIONING WITH MANUAL DATA INPUT mode of operation is particularly convenient for simple machining operations or exact pre-positioning of the electrode. You can write a program in conversational programming and execute it immediately. You can also define and call TNC cycles. The program is stored in the file $MDI.

- PGM CALL can not be used to call a program.
- LBL CALL can not be used for calling sub-routines or repeating sections of programs.
- For a TOOL CALL block to processed, the corresponding TOOL DEF tool definition must be programmed within the $MDI file.
- Incremental positionings always refers to the present electrode position.
- Programming a radius compensation (RL/RR) is not permitted.

Positioning with manual data input (MDI)

Select the Positioning with MDI mode of operation. Program the file $MDI as you wish.

I To start program run, press the machine START button.

Example: Programming and processing a line

Select operating mode: Positioning with MDI.

X Select the axis, and enter the end-point coordinates of the line and the feed rate, i.e.: **`L X+125 R F100 M`**

END Conclude entry.

I Start positioning block.

Protecting and erasing programs in $MDI

The $MDI file is generally intended for short programs that are only needed temporarily. Nevertheless, you can store a program, if necessary, by proceeding as described below:

Select the Programming and Editing mode of operation.

PGM MGT

To call the file manager, press the PGM MGT key (program management).

Move the highlight to the $MDI file.

COPY TNC ⇒ TNC

To select the file copying function, press the COPY soft key.

TARGET FILE =

74523 Enter the name under which you want to save the current contents of the $MDI file.

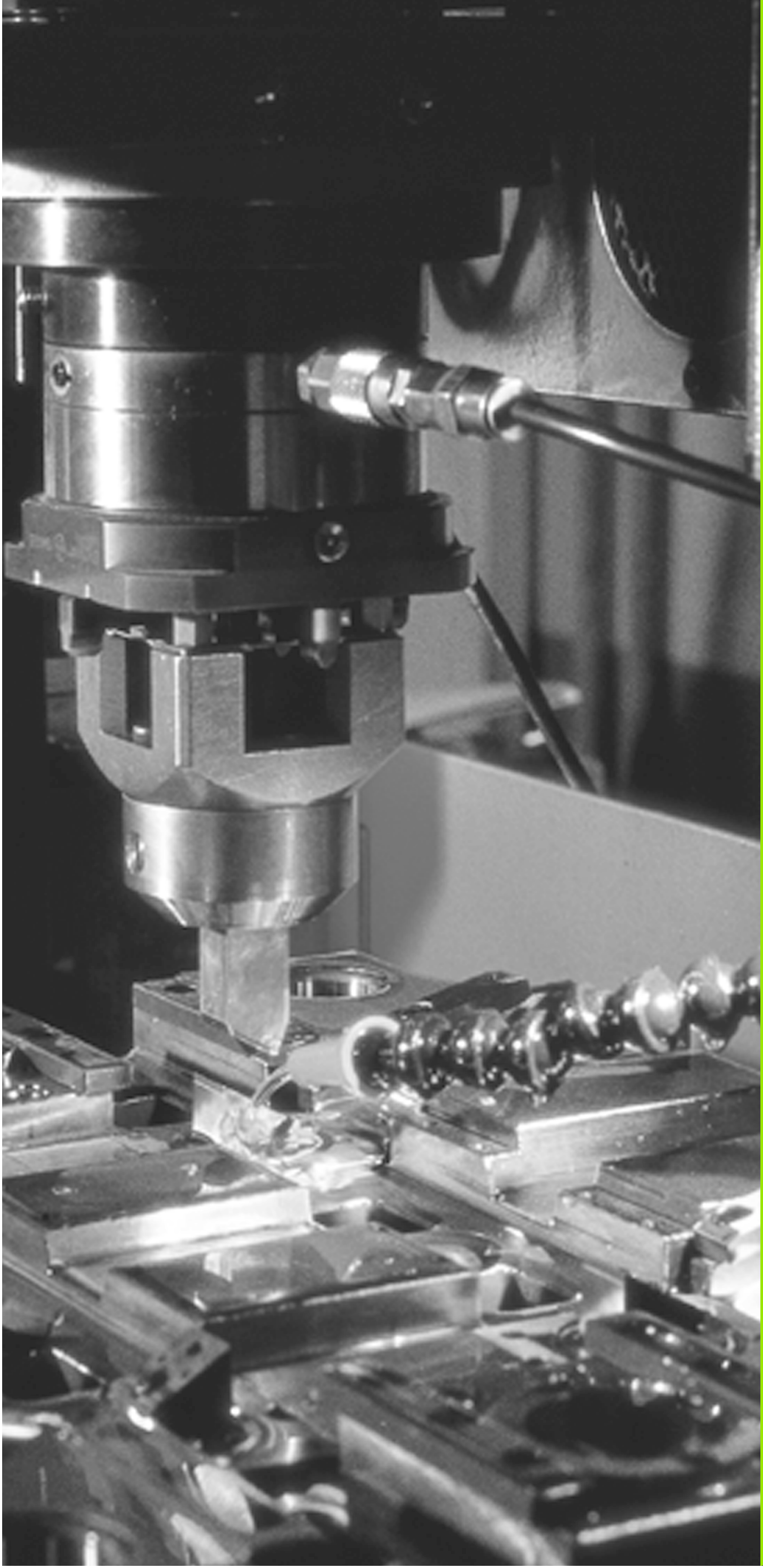
ENT

End the copying process with the ENT key.

Erasing the contents of the $MDI file is done in a similar way: Instead of copying the contents, however, you erase them with the DELETE soft key. The next time you select the operating mode Positioning with MDI, the TNC will display an empty $MDI file.

If you wish to delete $MDI, then

- you must not have selected the Positioning with MDI mode.
- you must not have selected the $MDI file in the Programming and Editing mode.



4

Programming: Fundamentals, Files, Program Entry, Spark Erosion, Erosion Tables

4.1 Fundamentals of Positioning

Introduction

This chapter covers the following topics:

- What is NC?
- The part program
- Programming
- Position encoders and reference marks
- Reference system
- Reference system with electrical discharge machines (EDM)
- Programming electrode movement
- Polar coordinates
- Absolute and incremental workpiece positions
- Setting the datum

What is NC?

NC stands for Numerical Control, that is, the operation of a machine tool by a series of coded instructions comprised of numbers.

Modern controls such as TNCs have a built-in computer for this purpose and are therefore called CNC (Computerized Numerical Control).

The part program

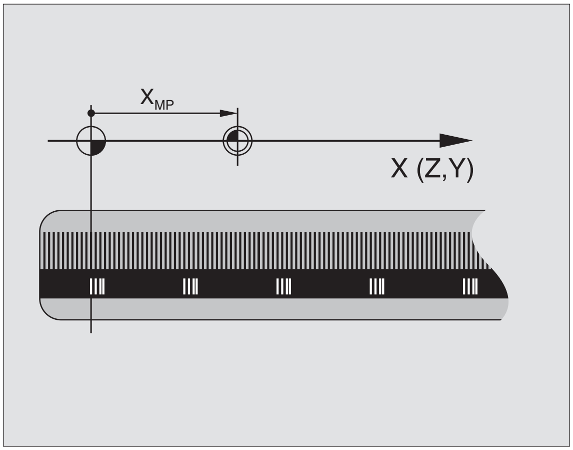
The part program is a complete list of instructions for machining a part. It contains such information as the target position of an electrode movement, the path function (how the electrode should move toward the target position) and the feed rate.
Information on the radius and length of the electrode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

Programming

Conversational programming is a particularly easy method of writing and editing part programs.
HEIDENHAIN NCs were developed specifically for the machine operator who keys in programs right at the machine. This is why they are called TNC (**T**ouch **N**umerical **C**ontrol).
You begin each machining step by pressing a key. The TNC then asks you for all the information it needs to execute the step.

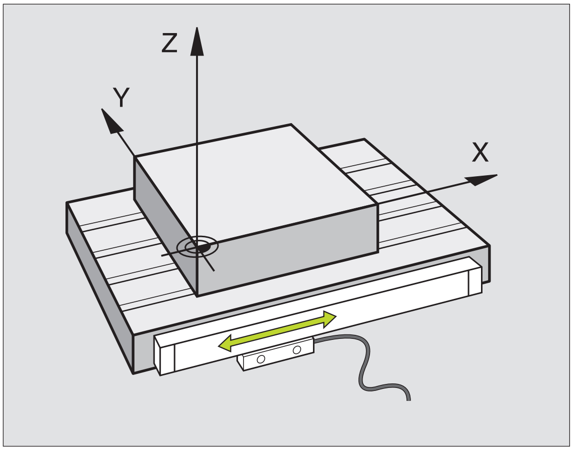
Position encoders and reference marks

The machine axes are equipped with position encoders that register the positions of the machine table or tool. When a machine axis moves, the corresponding position encoder generates an electrical signal. The TNC evaluates this signal and calculates the precise actual position of the machine axis.



If there is an interruption of power, the calculated position will no longer correspond to the actual position of the machine slide. The TNC can re-establish this relationship with the aid of reference marks when power is returned. The scales of the position encoders contain one or more reference marks that transmit a signal to the TNC when they are crossed over. From the signal the TNC identifies that position as the machine-axis reference point and can re-establish the assignment of displayed positions to machine axis positions.

Linear encoders are generally used for linear axes. Rotary tables and tilt axes have angle encoders. If the position encoders feature distance-coded reference marks, you only need to move each axis a maximum of 20 mm (0.8 in.) for linear encoders, and 20° for angle encoders, to re-establish the assignment of the displayed positions to machine axis positions.

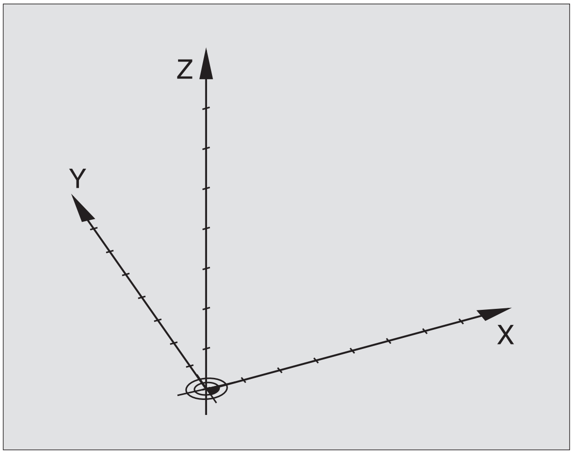


Reference system

A reference system is required to define positions in a plane or in space. The position data are always referenced to a predetermined point and are described through coordinates.

The Cartesian coordinate system (a rectangular coordinate system) is based on the three coordinate axes X, Y and Z. The axes are mutually perpendicular and intersect at one point called the datum. A coordinate identifies the distance from the datum in one of these directions. A position in a plane is thus described through two coordinates, and a position in space through three coordinates.

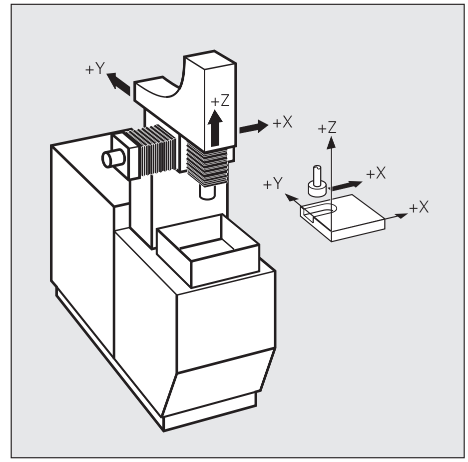
Coordinates that are referenced to the datum are referred to as absolute coordinates. Relative coordinates are referenced to any other known position (datum) you define within the coordinate system. Relative coordinate values are also referred to as incremental coordinate values.



Reference system with EDMs

When using an EDM, you orient tool movements to the Cartesian coordinate system. The illustrations at right show how the Cartesian coordinate system describes the machine axes. The figure at center 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.

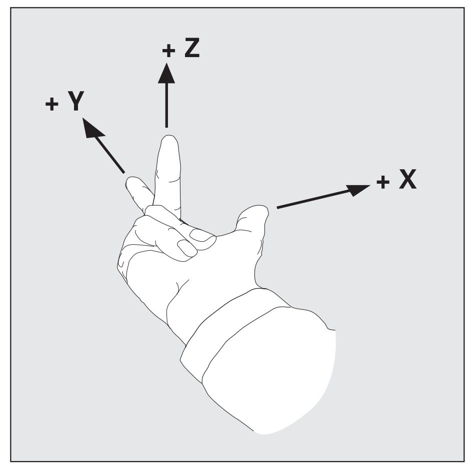
The TNC 406/TNC 416 can control up to 5 axes. The axes U, V and W are secondary linear axes parallel to the main axes X, Y and Z, respectively. Rotary axes are designated as A, B and C. The illustration at lower right shows the assignment of secondary axes and rotary axes to the main axes.



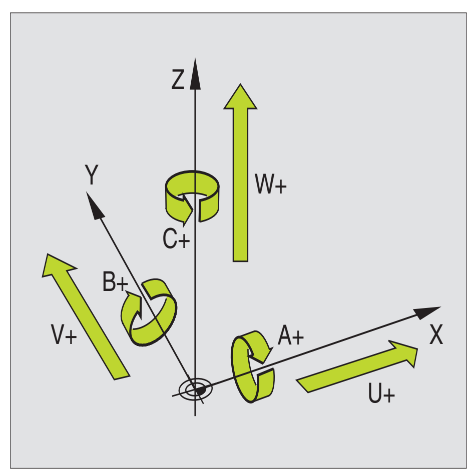
Programming electrode movement

Depending on the machine tool, either the machine table with the workpiece moves or the electrode moves.

> You always program as if the electrode moves and the workpiece remains stationary, no matter the type of machine.

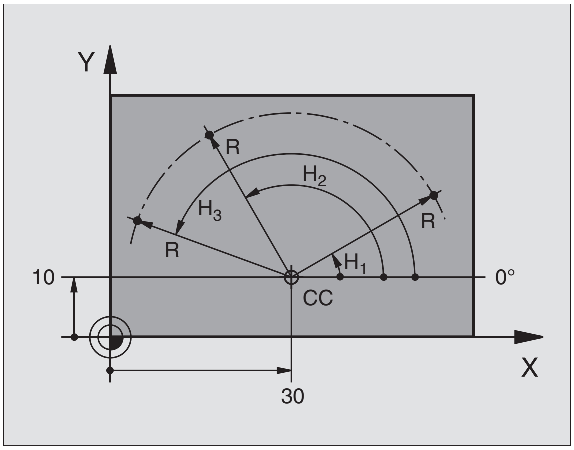


If the machine table moves, the corresponding axes are identified on the machine operating panel with a prime mark (e.g., X', Y'). The programmed direction of such axis movement always corresponds to the direction of electrode movement relative to the workpiece but in the opposite direction.



Polar coordinates

If the production drawing is dimensioned in Cartesian coordinates, you also write the part program using Cartesian coordinates. For parts containing circular arcs or angles it is often simpler to give the dimensions in polar coordinates (see „Path Contours — Polar Coordinates" on page 109).



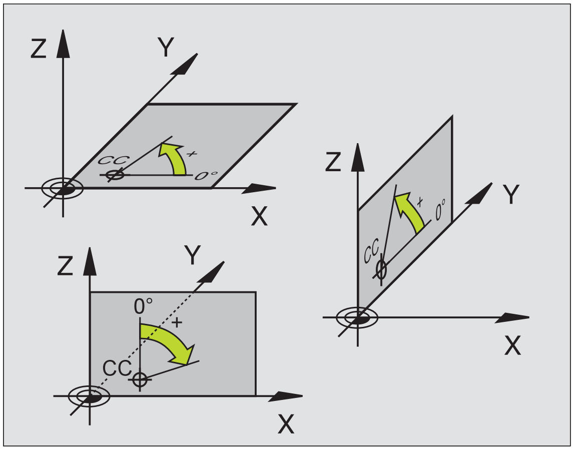
While the Cartesian coordinates X, Y and Z are three-dimensional and can describe points in space, polar coordinates are two-dimensional and describe points in a plane. Polar coordinates have their datum at a circle center (CC), or pole. A position in a plane can be clearly defined by the:

- Polar Radius, the distance from the circle center CC to the position, and the
- Polar Angle, the size of the angle between the reference axis and the line that connects the circle center CC with the position.

See figure at upper right.

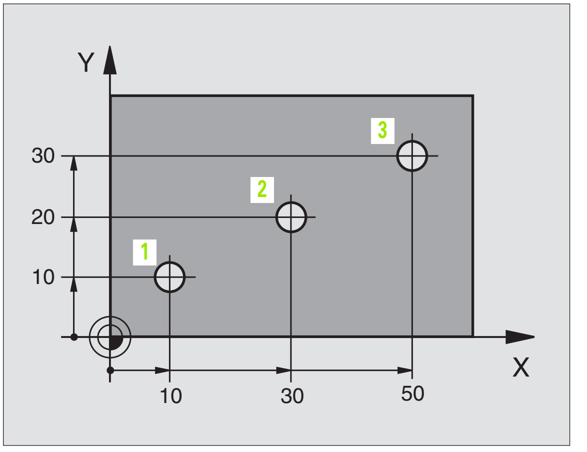
Definition of pole and angle reference axis

The pole is set by entering two Cartesian coordinates in one of the three planes. These coordinates also set the reference axis for the polar angle PA.



Coordinates of the pole (plane)	Reference axis of the angle
X/Y	+X
Y/Z	+Y
Z/X	+Z

Absolute and incremental workpiece positions

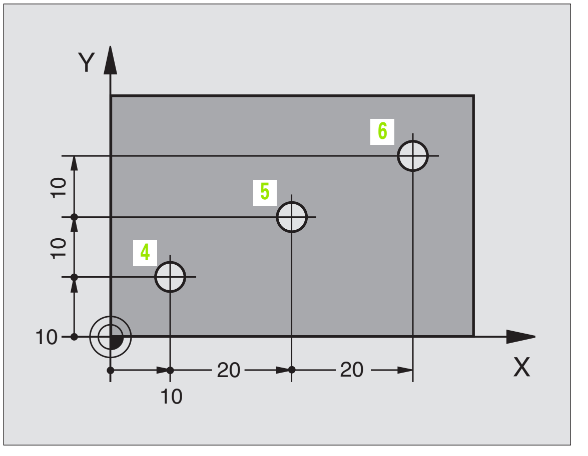


Absolute workpiece positions

Absolute coordinates are position coordinates that are referenced to the datum of the coordinate system (origin). Each position on the workpiece is uniquely defined by its absolute coordinates.

Example 1: Holes dimensioned in absolute coordinates

Hole **1**	Hole **2**	Hole **3**
X = 10 mm	X = 30 mm	X = 50 mm
Y = 10 mm	Y = 20 mm	Y = 30 mm



Incremental workpiece positions

Incremental coordinates are referenced to the last programmed nominal position of the tool, which serves as the relative (imaginary) datum. When you write a part program in incremental coordinates, you thus program the tool to move by the distance between the previous and the subsequent nominal positions. Incremental coordinates are therefore also referred to as chain dimensions.

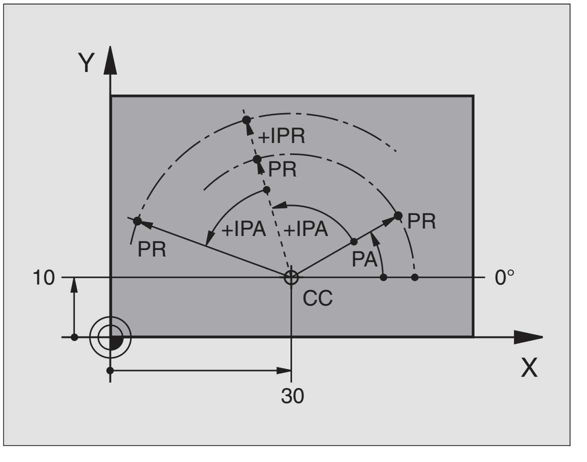
To program a position in incremental coordinates, enter the prefix "I" before the axis.

Example 2: Holes dimensioned in incremental coordinates

Absolute coordinates of hole **4**

X = 10 mm
Y = 10 mm

Hole **5**, referenced to **4**	Hole **6**, referenced to **5**
X = 20 mm	X = 20 mm
Y = 10 mm	Y = 10 mm



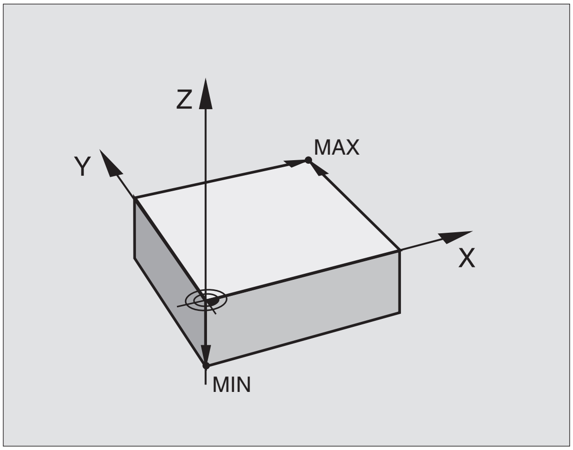
Absolute and incremental polar coordinates

Absolute polar coordinates always refer to the pole and the reference axis.

Incremental polar coordinates always refer to the last programmed nominal position of the tool.

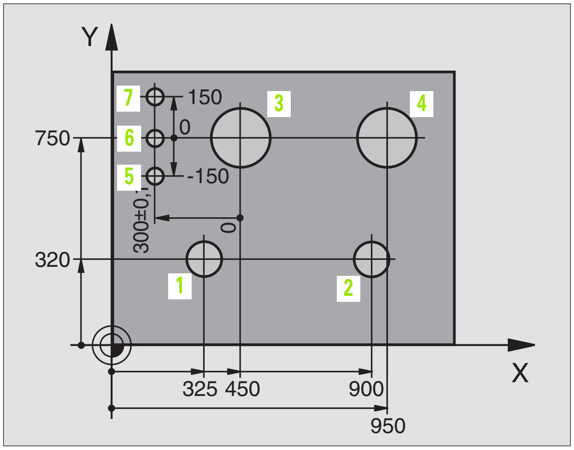
Setting the datum

The production drawing identifies a certain form element of the workpiece (usually a corner) as the absolute datum, and usually one or more form elements as relative datums. Through the datum setting process, the origin of the absolute or relative coordinate systems is set to these datums: The workpiece – aligned to the machine axes – is brought into a certain position relative to the electrode, and the display is set to zero or the appropriate position value (i.e., to account for the electrode radius) (see „Datum Setting" on page 22).



Example

The workpiece drawing at right shows holes (**1** to **4**) whose dimensions are shown with respect to an absolute datum with the coordinates X=0, Y=0. The holes (**5** to **7**) are dimensioned with respect to a relative datum with the absolute coordinates X=450, Y=750. With the **DATUM SHIFT** cycle you can temporarily set the datum to the position X=450, Y=750, to be able to program the holes (**5** to **7**) without further calculations.



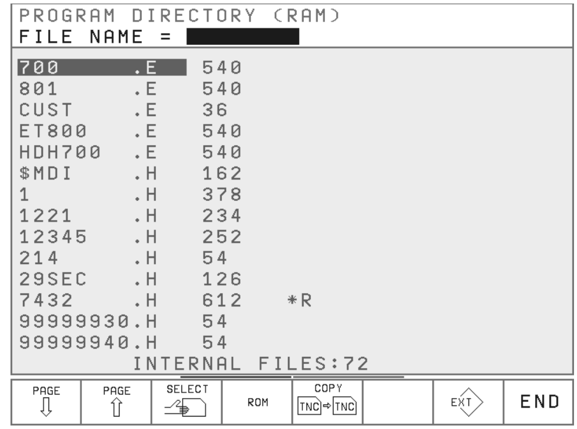
4.2 Files

The TNC 416 saves programs and tables as files. The TNC can store up to 100 files. A file is identified by its file name and file extension.

The file name is entered when a new file is created.

The file extension is separated from the file name by a period, and indicates what type of file it is.

Files in the TNC	Type
Programs In HEIDENHAIN format	 .H
Tables for Erosion Datum Tools Time capture	 .E .D .T Time.W



The tool table TOOL.T is only active if bit 2 of MP7224 is set to 0.

File directory

You call the file directory with the PGM NAME key (TNC 406) or the PGM MGT key (TNC 416).

To delete files from the TNC, use CL PGM on the TNC 406 to call up the directory.

Overview of the file management functions:

File	Operating mode	Call file directory with
... Create		PGM NAME or PGM MGT
... Edit		PGM NAME or PGM MGT
... Delete		CL PGM or PGM MGT
... Test		PGM NAME or PGM MGT
... Run		PGM NAME or PGM MGT

The file directory contains the following information:

Display	Meaning
FILE NAME	Name (up to 8 characters plus file extension)
BYTE	File size in bytes
STATUS R E P I	Properties of the file: File is active for Program Run/Program Test. File is active for Programming and Editing. File is protected against editing and erasure. Dimensions are given in inches.
STORAGE AREA AND NUMBER INTERNAL FILES	 Files in the TNC memory
EXTERNAL FILES	Files, e.g., on a PC 401

Files in ROM

Pressing the ROM soft key displays files that the machine tool builder wrote and stored in ROM, such as erosion tables. These files can be edited.

Identification of protected files

The TNC inserts a ”P” in the first and last lines of write- and erase-protected files.
The file directory also shows a ”P” next to the file name.

Selecting, copying, deleting and protecting files

Activate the file directory

- Use the PGM MGT key with the TNC 416, and the PGN NAME key with the TNC 406. If you want to delete files with the TNC 406, you must call the file directory with the CL PGM key.

Select the file

- Enter the file name (not for CL PGM) or move the highlight with the cursor keys to the desired file.

Function	Soft key
Go to the next page	PAGE ⇩
Go to the previous page	PAGE ⇧
Display files in ROM	ROM
Select file (such as for a test run)	SELECT
Copy file: Enter the name of the target file	COPY TNC → TNC
File protection	PROTECT
Cancel file protection	UNPROTECT
Deleting a file	DELETE
Close the file directory	END

4.3 Creating and Writing Programs

Organization of an NC program in HEIDENHAIN conversational format.

A part program consists of a series of program blocks. The figure at right illustrates the elements of a block.

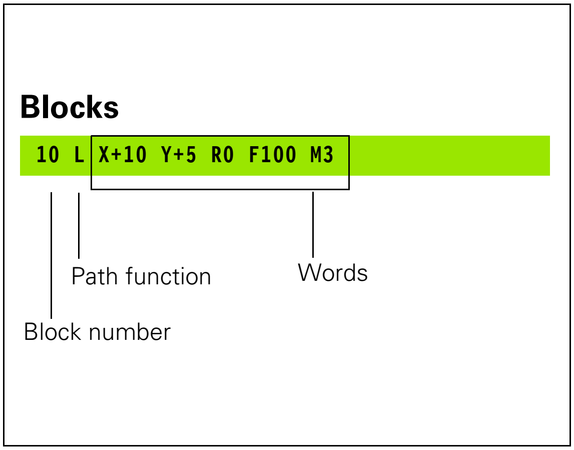
The TNC numbers the blocks in ascending sequence.

The first block of a program is identified by **BEGIN PGM**, the program name and the active unit of measure.

The subsequent blocks contain information on:

- The workpiece blank
- Tool definitions, tool calls
- Feed rates and spindle speeds, as well as
- Path contours, cycles and other functions

The last block of a program is identified by **END PGM**, the program name and the active unit of measure.

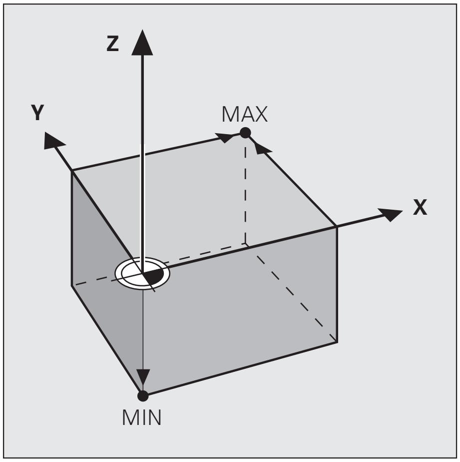


Defining the blank form–BLK FORM

Immediately after initiating a new program, you define a cuboid workpiece blank. If you wish to define the blank at a later stage, press the BLK FORM soft key. This definition is needed for the TNC's graphic simulation feature. The sides of the workpiece blank lie parallel to the X, Y and Z axes and can be up to 30 000 mm long. The blank form is defined by two of its corner points:

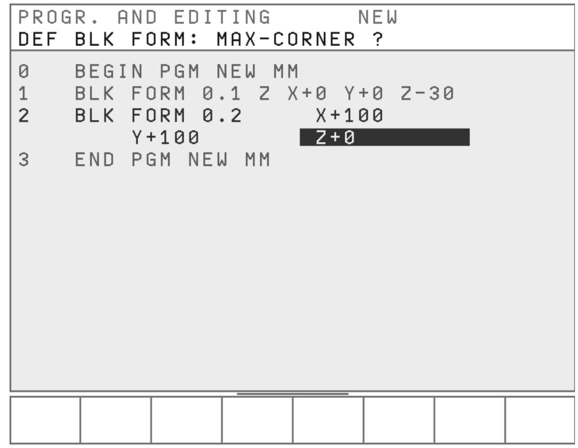
- MIN point: the smallest X, Y and Z coordinates of the blank form, entered as absolute values.
- MAX point: the largest X, Y and Z coordinates of the blank form, entered as absolute or incremental values.

> You only need to define the blank form if you wish to run a graphic test for the program!



Creating a new part program

You always enter a part program in the **Programming and Editing** mode of operation. Program initiation in an example:



Select the **Programming and Editing** mode of operation.

PGM NAME

Press the key to call the file directory.

FILE NAME = 7432

ENT

Enter the new program name and confirm your entry with the ENT key.

.H

Choose the type of file: Press the .H, .E or .D soft key. The TNC changes to the program window.

BLK FORM

To define the BLK-FORM, press the BLK-FORM soft key. The TNC opens a dialog for defining the **BLK FORM.**

WORKING SPINDLE AXIS X/Y/Z ?

Enter the spindle axis.

DEF BLK FORM: MIN-CORNER ?

0 ENT

0 ENT

-40 ENT

Enter in sequence the X, Y and Z coordinates of the MIN point.

DEF BLK FORM: MAX-CORNER ?

100 ENT

100 ENT

0 ENT

Enter in sequence the X, Y and Z coordinates of the MAX point.

Example: Display the BLK form in the NC program.

`0 BEGIN PGM NEW MM`	Program begin, name, unit of measure
`1 BLK FORM 0.1 Z X+0 Y+0 Z-40`	Spindle axis, MIN point coordinates
`2 BLK FORM 0.2 X+100 Y+100 Z+0`	MAX point coordinates
`3 END PGM NEW MM`	Program end, name, unit of measure

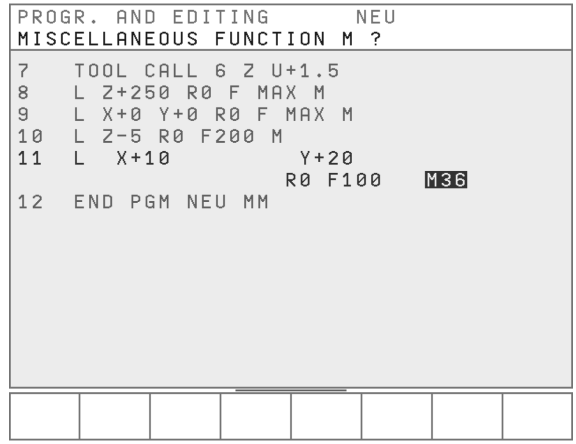
The TNC automatically generates the block numbers as well as the **BEGIN** and **END** blocks.

If you do not wish to define a blank form, cancel the dialog at **`Working spindle axis X/Y/Z`** by pressing the DEL key!

The TNC can display the graphic only if the ratio of the short side to the long sides of the **`BLK FORM`** is greater than 1:64!

Programming tool movements in conversational format

To program a block, initiate the dialog by pressing a function key. In the screen headline, the TNC then asks you for all the information necessary to program the desired function.



Example of a dialog

Dialog initiation

COORDINATES ?

X **10** Enter the target coordinate for the X axis.

Y **20** ENT Enter the target coordinate for the Y axis, and go to the next question with ENT.

RADIUS COMP. RL/RR/NO COMP. ?

ENT Enter "No radius compensation" and go to the next question with ENT.

FEED RATE ? F= / F MAX = ENT

100 ENT Enter a feed rate of 100 mm/min for this path contour; go to the next question with ENT.

MISCELLANEOUS FUNCTION M ?

36 ENT Enter the miscellaneous function **M36** "eroding ON"; pressing the ENT key terminates this dialog.

The program blocks window will display the following line:

3 L X+10 Y+20 R0 F100 M36

Function	Key
Continue the dialog	ENT
Ignore the dialog question	NO ENT
End the dialog immediately	END
Abort the dialog and erase the block	DEL

Editing a program

While you are creating or editing a part program, you can select any desired line in the program or individual words in a block with the arrow keys or the soft keys:

Function	Soft keys/keys
Go to the previous page	PAGE ⇧
Go to the next page	PAGE ⇩
Move from one block to the next	↓ ↑
Select individual words in a block	← →

Function	Key
Set the selected word to zero	CE
Erase an incorrect number	CE
Clear a (non-blinking) error message	CE
Delete the selected word	NO ENT
Delete the selected block	DEL
Erase cycles and program sections: First select the last block of the cycle or program section to be erased, then erase with the DEL key.	DEL

Inserting blocks at any desired location

- Select the block after which you want to insert a new block and initiate the dialog.

Editing and inserting words

- Select a word in a block and overwrite it with the new one. The plain-language dialog is available while the word is highlighted.
- To accept the change, press the END key.

If you want to insert a word, press the horizontal arrow key repeatedly until the desired dialog appears. You can then enter the desired value.

Looking for the same words in different blocks

→	To select a word in a block, press the arrow keys repeatedly until the highlight is on the desired word.
↓	Select a block with the arrow keys.
GOTO □	Select a block directly.

The word that is highlighted in the new block is the same as the one you selected previously.

4.4 Automatic Workpiece Change with WP-Call

If your machine features an automatic handling system, you can program an automatic workpiece change with the WP-CALL function. WP-CALL resets an active rotation, and can be programmed to subsequently execute a datum shift and activate the rotation again, if desired. The values for datum shift and rotation are transferred by the PLC.

The function for automatic workpiece change is adapted to the TNC by the machine tool builder. Refer to your machine tool manual.

Programming a workpiece change

- Select the Programming and Editing mode of operation.

WP-CALL

- Press the WP-CALL soft key.
- **Workpiece name:** Enter the name of the pallet (for example, 1). You can enter up to 16 characters (letters and numbers).
- **Number of tilts:** Enter the number of tilts (maximum input value: 9).

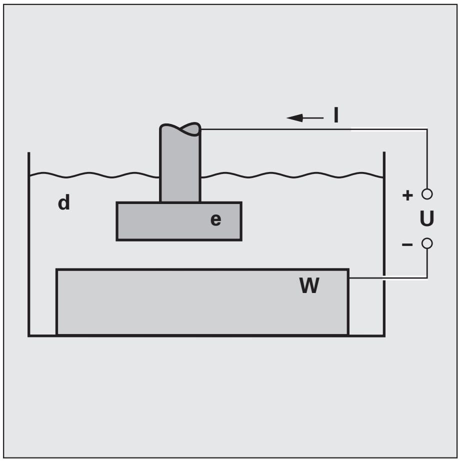
Example NC block

```
7  WP-CALL 1            / 1
```

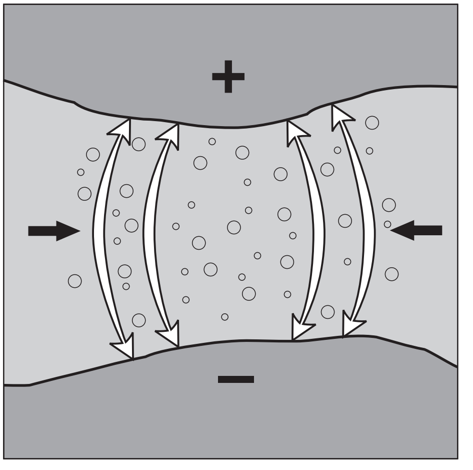
4.5 Fundamentals of Spark Erosion

Electrical discharge machining is an electrothermal process which uses a spark to remove metal by melting and vaporizing (”eroding”) the workpiece surface.
In contrast, cutting machine tools such as milling machines remove metal by direct abrasive action.
The spark erosion process is described on the following pages.

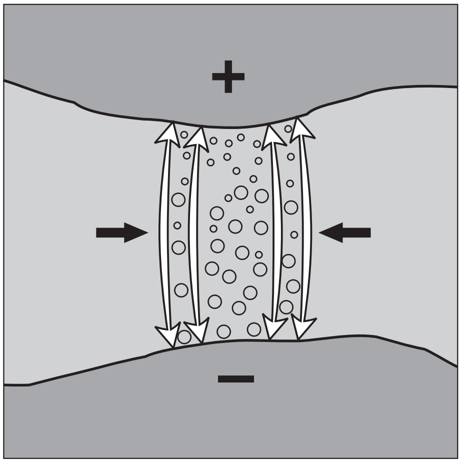
The electrode (e) and the workpiece (w) are submerged in a dielectric fluid (d).
A generator applies a voltage to the electrode and the workpiece (both the electrode and the workpiece are then electrodes).



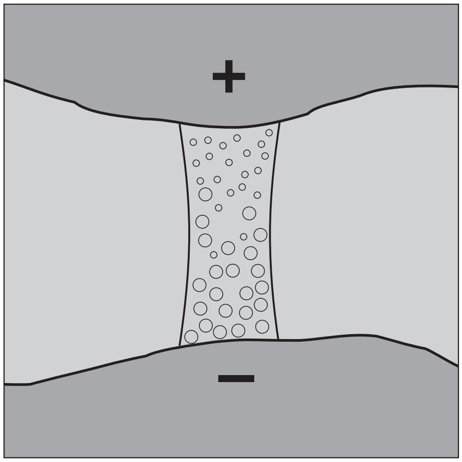
An electric field is then created in the gap between the electrode and the workpiece.
The electric field is strongest where the gap is the smallest.
The electrically conductive particles in the dielectric fluid are concentrated at this point.



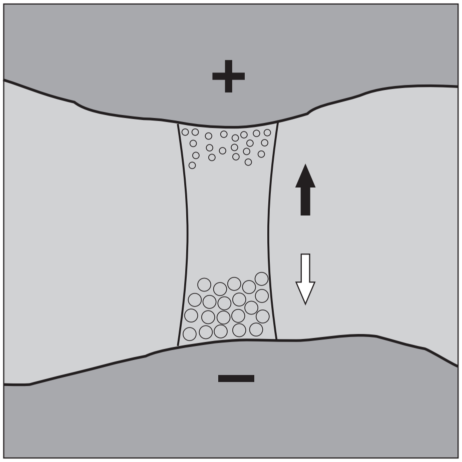
A bridge of electrically conductive particles forms between the electrode and the workpiece.



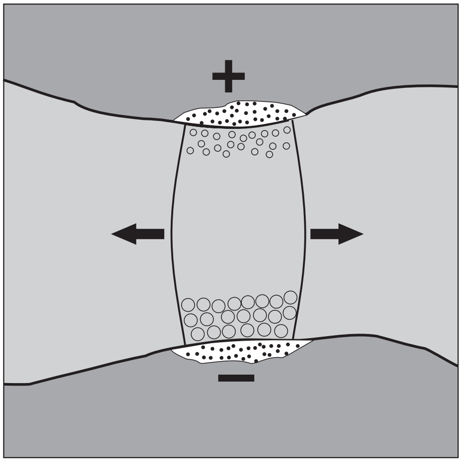
After a certain length of time (the ignition delay time), a discharge channel suddenly forms across the bridge of particles, and current starts to flow between the electrode and the workpiece. The current flow increases the temperature in the discharge channel, and further electrically charged particles are created (ions). The current increases.



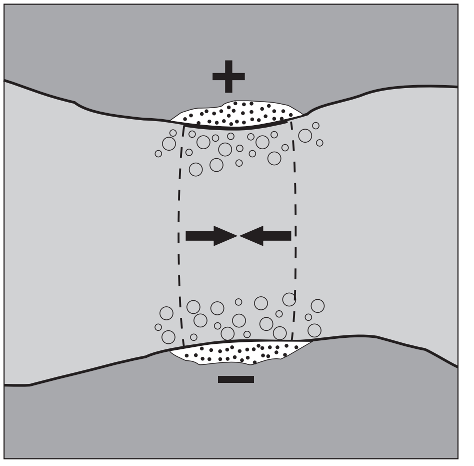
The temperature in the discharge channel becomes so great that the dielectric fluid there vaporizes.



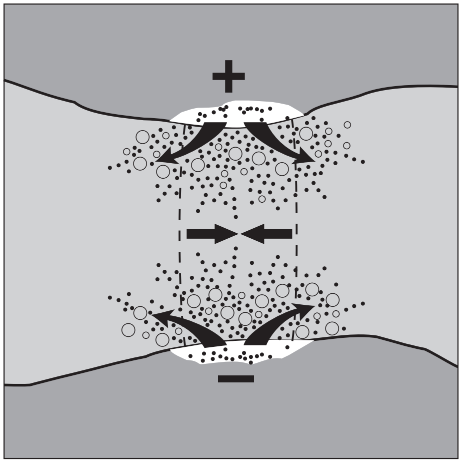
The discharge channel expands in the middle while at the electrode and workpiece it becomes narrower.
The temperature increases to a point where the surfaces of the electrode and workpiece melt. Part of the molten metal vaporizes.



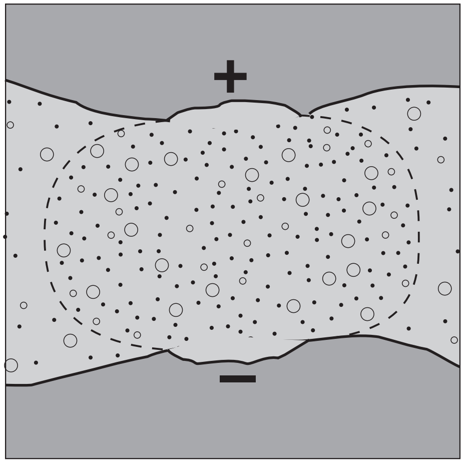
When the voltage is removed, the discharge channel collapses (implodes).



When the discharge channel collapses, the implosion thrusts the molten metal into the dielectric fluid.



A small crater remains on the electrode and the workpiece. The debris of melted electrode or workpiece material remains suspended in the dielectric fluid.



4.6 Erosion Tables

The machine tool builder can define the erosion tables as required. He may also define additional parameters that are not mentioned in your TNC manual. Refer to your machine tool manual.

The spark erosion process is influenced by process variables called erosion parameters. You can enter the erosion parameters for a machining sequence in erosion tables for the TNC 406/416.
For example, you can create a separate erosion table for each combination of electrode and workpiece material.
All parameters are then clearly grouped in this table. The TNC can access the parameters for a particular machining sequence.

Using erosion tables in a program

If you want to work with erosion tables in a program, you must copy Cycle 1 (GENERATOR) into the program (see „Cycle 1 GENERATOR" on page 133). In this cycle you declare what erosion table you are working with.

Working without an erosion table

It is also possible to work without an erosion table. In this case the TNC stores the erosion parameters in the Q parameters Q90 to Q99 (see „Preassigned Q parameters" on page 202). Your machine manual provides more information on these Q parameters.

Ready-to-use erosion tables

The machine builder can prepare erosion tables and store them in the TNC's ROM. Proceed as follows if you want to work with these erosion tables:

- Press the PGM NAME key in the PROGRAMMING AND EDITING mode of operation.
- Press the ROM soft key.

The machine tool builder can give you additional information on these erosion tables.

4.7 Parameters in the Erosion Table

You can enter the following erosion parameters in one erosion table:

Meaning	Range
Power stage (NR)	25 to 1
Low voltage current (LV)	0 to 99
High voltage current (HV)	0 to 9
Gap voltage (GV)	0 to 99
Pulse-on duration (TON)	0 to 999
Pulse-off duration (TOFF)	0 to 255
Servo sensitivity (SV)	0 to 99 %
Auto jump distance (AJD)	0 to 99.9 mm
Erosion time (ET)	0 to 999 s
Arc sensitivity (AR)	0 to 99
Electrode polarity (P)	0 or 1
High voltage selector (HS)	0 to 99
Wear rate (WR)	0 to 99 %
Surface finish (RA)	0 to 99.9 µm
Stock removal (SR)	0 to 999.999 ccm/min
Two-times gap (2G)	0 to 9.999 mm
Minimum undersize (UNS)	0 to 9.999 mm
Auxiliary parameters (AUX 1 to AUX 6)	

To enter erosion parameters in the erosion table

PGM NAME	Activate file management.

FILE NAME ?

↓ ↑	Select a file with the arrow keys.
15	Enter the file name directly, e.g. 15. For a new erosion table you must enter the name.
ENT	Open the selected erosion table.

PROGRAM SELECTION

.E	Select program type .E (erosion table).

POWER STAGE NUMBER

1 ENT	Enter the number of the power stage for the following data. Confirm with ENT.

The TNC then asks for all further erosion parameters described in this chapter.

To enter erosion parameters for additional power stages

INSERT	With INSERT, erosion parameters for up to 25 power stages can be entered.

To conclude entry

Return to program management with PGM NAME.

To go to a certain power stage

Use GOTO to directly access a certain power stage number in the erosion table (do not enter the table row number).

Unit of measurement in the table

With the TAB soft key you can change the name of the table and the unit of measurement. The same unit (millimeters or inches) should be used in the erosion tables as in the NC program.

Power stage (NR)

The power stages determine the type of machining (roughing, finishing or polishing).

Recommended input

- Roughing NR = 15 to 10
- Finishing NR = 10 to 6
- Fine finishing NR = 6 to 1
- Polishing NR = 5

Input range

15 (25) to 1 in decreasing order.

To change the power stage in the program

The current power stage is given by Q parameter Q99.
If you change Q99, you also change the power stage.

Low voltage current (LV)

The machine tool builder can give you information on this erosion parameter. Refer to your machine tool manual.

Input range

0 to 99 in up to 100 increments.

High voltage current (HV)

The machine tool builder can give you information on this erosion parameter. Refer to your machine tool manual.

Input range

0 to 9 in up to 10 increments.

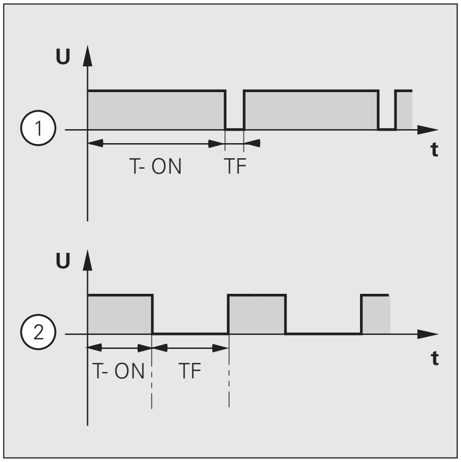
Gap voltage (GV)

The TNC adjusts the width of the gap between the electrode and the workpiece by controlling the gap voltage. The nominal gap voltage GV should be chosen with care.

Setting

- If the gap voltage is too high, the rate of stock removal will be too low.
- If the gap voltage is too low, irregularities will occur (arcing, short circuiting).

Pulse-on duration and pulse-off duration



The **pulse-on duration (TON)** is the time in which the generator applies a voltage to the electrode and workpiece.
Ignition and subsequent discharge take place during this time.

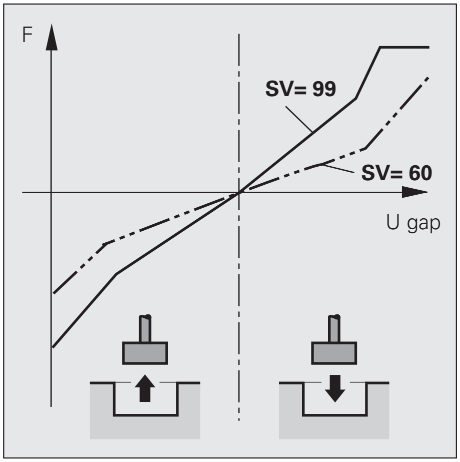
The **pulse-off duration (TOF)** is the time in which no voltage is generated.
During this time the gap is flushed and deionized.

Select the TON/TOF ratio according to the type of machining:

Setting

- Roughing: Long pulse-on duration, short pulse-off duration
- Finishing and polishing: Short pulse-on duration, long pulse-off duration

Servo sensitivity SV



The machine tool builder specifies a characteristic curve for servo sensitivity (see figure center right). Refer to your machine tool manual.

The servo sensitivity influences the reaction speed of the gap control.

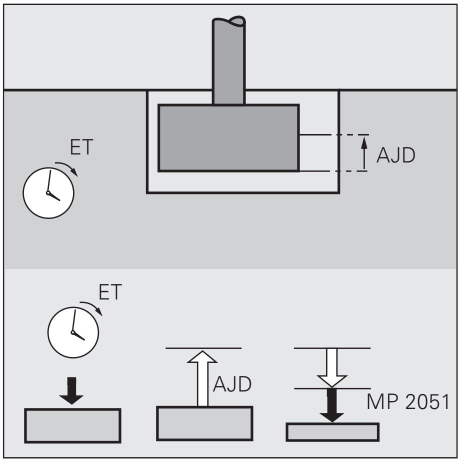
Setting

- High servo sensitivity: fast gap control
- Low servo sensitivity: slow gap control

Input range

0 to 99 %

Erosion time ET, Auto jump distance AJD



The **erosion time** determines how long an erosion step lasts. When the programmed erosion time has run out, the electrode retracts by the **auto jump distance** and subsequently returns to the position given in machine parameter MP2051.

Intermittent flushing

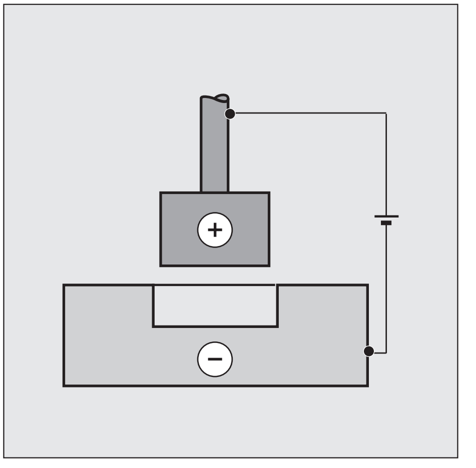
To improve deionization of the gap and flush away debris, you can activate miscellaneous function M8 (intermittent flushing ON).

Arc sensitivity (AR)

The arc sensitivity setting influences the gap signal that the generator sends to the TNC. The characteristic curve shows the nominal speed value plotted against the gap voltage.

The machine tool builder can give you information on this erosion parameter. Refer to your machine tool manual.

Electrode polarity (P)



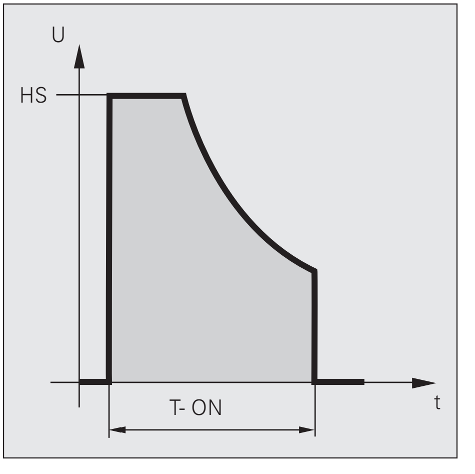
To minimize wear on the electrode and ensure a high rate of stock removal, you must set the correct electrode polarity.

Input value

- Positive electrode: 0
- Negative electrode: 1

If you mount the electrode on the machine table, you must change the electrode polarity defined in the machine table. The TNC does not reverse the polarity automatically.

High voltage selector HS



The high voltage is the voltage that the generator applies to the electrode and workpiece.

Setting

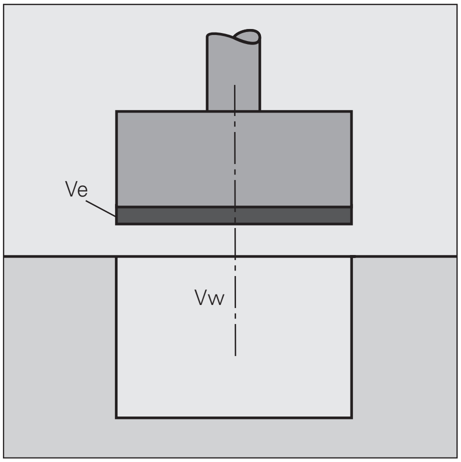
- High value for HS:
  For large gaps and for high rate of stock removal.
- Low value for HS (with ignition pulse):
  For small gaps and for low rate of stock removal.
- Low value for HS (without ignition pulse):
  For a few specific hard metals and very small electrodes.

Wear rate WR

The wear rate is the ratio between the volume of material removed from an electrode (Ve) and the volume of material removed from the workpiece (Vw).

WR = Ve / Vw • 100 %

For the wear rate on the electrode for your particular machining task and combination of materials, refer to the electrode table.



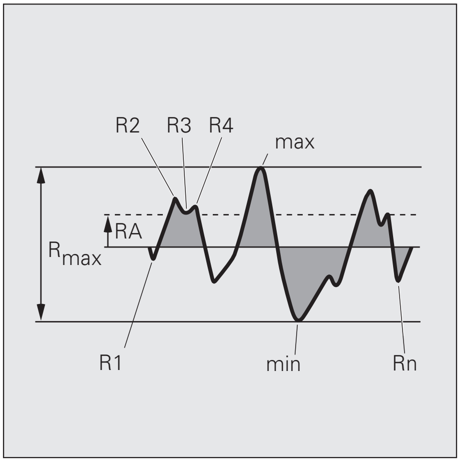
Surface finish RA

Surface finish is a measure of machining quality. A machined surface is never absolutely smooth, but consists of a series of peaks and valleys.

Maximum surface roughness R_{max}

The maximum surface roughness Rmax is the difference in height between the highest peak and the lowest valley.
The maximum surface roughness Rmax is also calculated from the width of the two-times gap 2G and the minimum undersize UNS as follows:

R_{max} = 0.5 • (UNS – 2G)

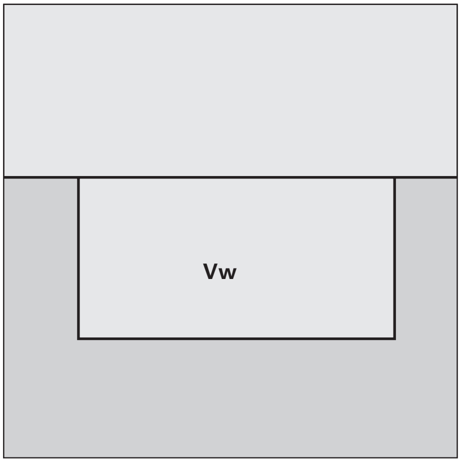


Determining surface finish RA according to VDI 3400

1. Determine the centerline of R_{max}
2. Measure all peaks and valleys from the centerline
3. Add the measured values together and divide by the number of measured values. The result is the surface finish RA in [µm]

Stock removal SR

The stock removal is the volume of removed workpiece material (Vw) per unit of time.
Stock removal is measured in ccm/minute.

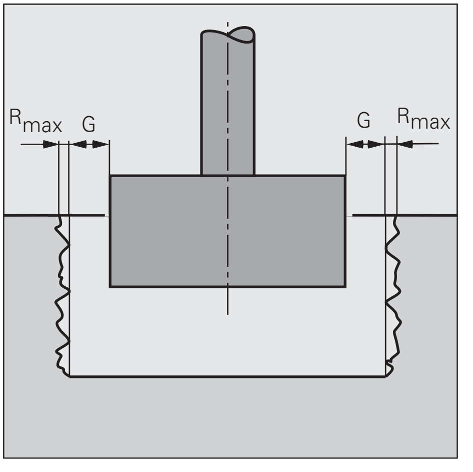


Two-times gap (2G)

During the erosion process, a minimum gap G must be maintained between the electrode and the workpiece. The higher the current, the larger the gap (G = radial gap) can (and should) be.

Minimum for the two-times gap

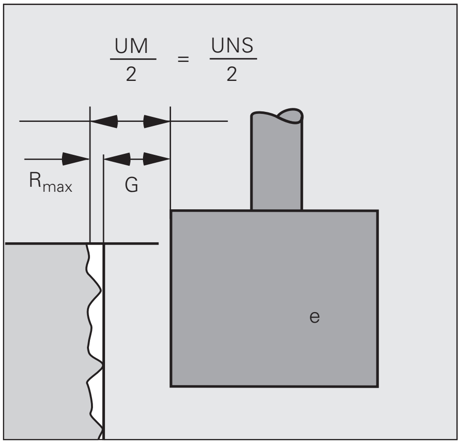
The two-times gap 2G is the minimum total gap (2 x G in millimeters) that must be maintained in the cavity between the electrode and the workpiece (2G = diametrical gap).



Minimum undersize (UNS)

The electrode diameter (Re) must be smaller than the cavity diameter by at least the value of the minimum undersize UNS.

- Roughing
  For roughing, the minimum undersize UNS is calculated from the two-times gap 2G and the maximum surface roughness R_{max}.
- Finishing and polishing
  For finishing and polishing, the minimum undersize UNS is equal to the two-times gap 2G. (The maximum surface roughness R_{max} can be disregarded.)

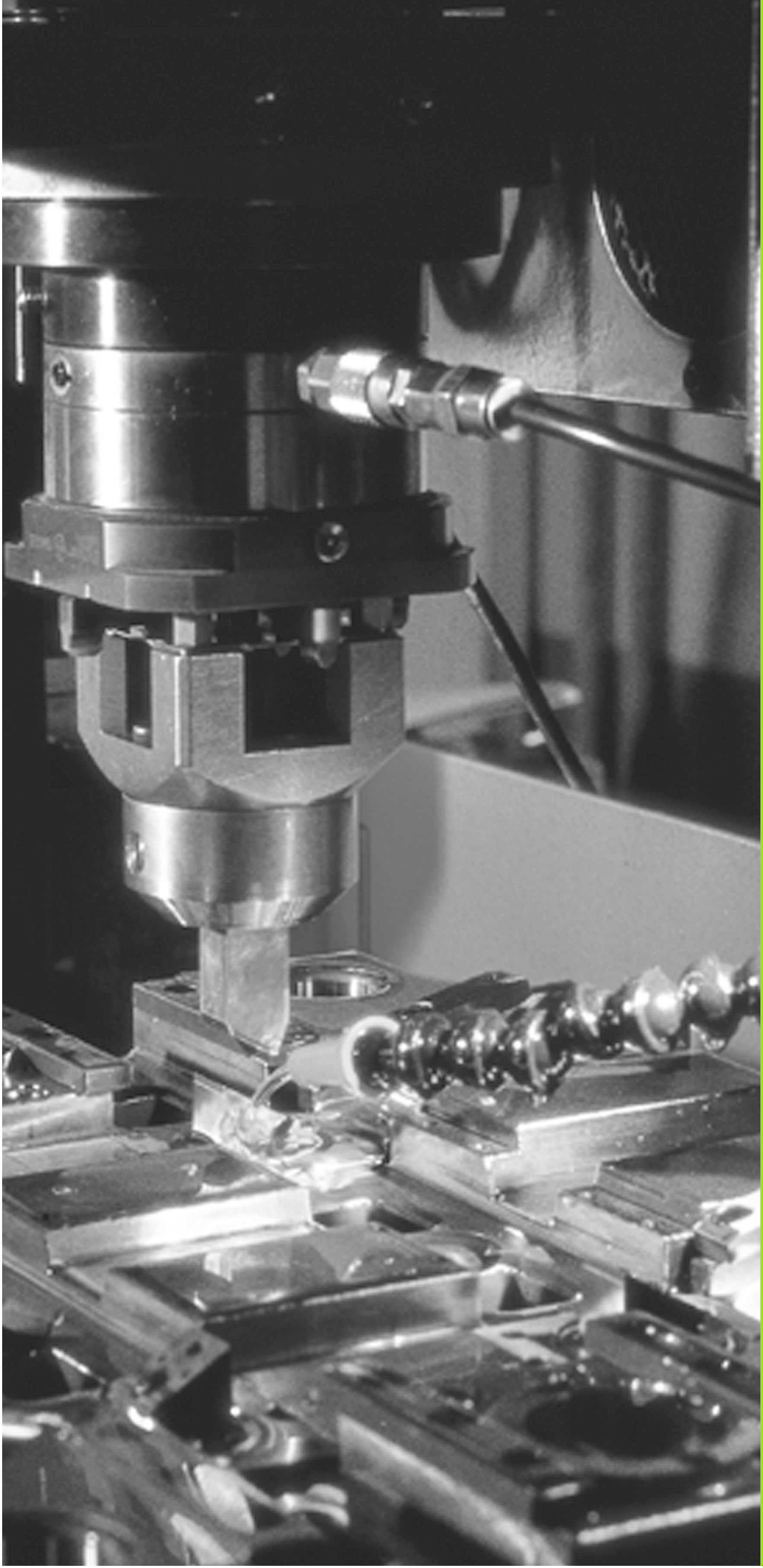


Selecting the actual undersize UM

- For a simple cavity (movement only in the electrode axis):
  UNS = UM
- For contour eroding and eroding with DISC cycle (movement of the electrode in all axes): UM ≥ UNS

Auxiliary parameters AUX 1, AUX 2, ... AUX 6

The machine tool builder can assign functions to up to six auxiliary parameters. Refer to your machine tool manual.



5

Programming: Tools

5.1 Electrodes

Each electrode is identified by a number.
The electrode data, consisting of the

- Length L
- Radius R

are assigned to the electrode number.

The electrode data are entered into the program with the TOOL DEF command.
The TNC takes the electrode length and radius into account when the electrode is called by its number.

If you are working with standard electrodes you can also define all the electrode data in a separate program.
In the part program you then call the program containing the electrode definitions with the PGM CALL command.

Electrode axis C

You can define the C axis as the electrode axis.
The TNC then operates as if the Z axis were the electrode axis. This also holds for radius compensation and for the ROTATION cycle.

Determining the electrode data

Electrode number

Each electrode is assigned a number from 0 to 99 999 999.

Electrode number 0 is defined as having length L = 0 and radius R = 0 when the electrode data are entered into the program.

Electrode radius R

The radius of the electrode is entered directly.

Electrode length L

The compensation value for the electrode length is defined

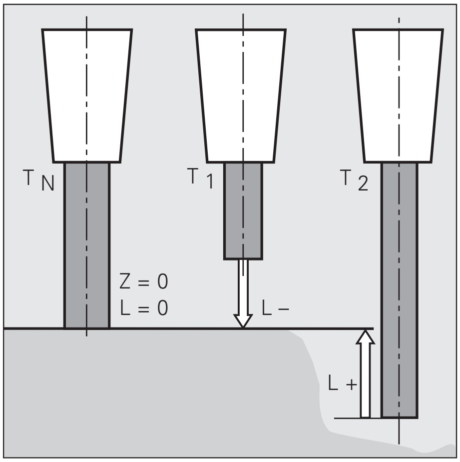
- as a length difference between the electrode and a zero electrode, or
- with a tool presetter.

If electrode lengths are determined with a tool presetter they should be entered directly into the electrode definition (TOOL DEF block) without further conversions.

Determining the electrode length with a zero electrode

Sign of the electrode length L:

L>L0:	The tool is longer than the zero tool
L<L0:	The tool is shorter than the zero tool



To determine the length:

- Move zero electrode to the reference position in electrode axis (such as workpiece surface with Z = 0).
- If necessary, set datum in electrode axis to zero.
- Insert electrode.
- Move electrode to the same reference position as zero electrode.
- The compensation value for length L of the electrode is displayed.
- Write down the value and enter it later, or transfer the value with the actual position capture function.

Entering electrode data into a program

For each electrode the electrode data can be entered once in the part program:

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

To enter the electrode data into a program block

The number, length and radius of a specific electrode is defined in the TOOL DEF block of the part program.

- To select tool definition, press the TOOL DEF key.

TOOL DEF

- Tool number : Assign a number to the electrode
- Tool length : Compensation value for the tool length
- Tool radius : Compensation value for the tool radius

- The electrode length L can be transferred directly into the electrode definition with the actual position capture function (see ”Actual Position Capture” on page 84).
- Cycle 3 TOOL DEF (see ”Cycle 3 TOOL DEF” on page 135) deletes the tool length from the TOOL DEF(inition)!

Example

```
4 TOOL DEF 5 L+10 R+5
```

Entering electrode data in tables

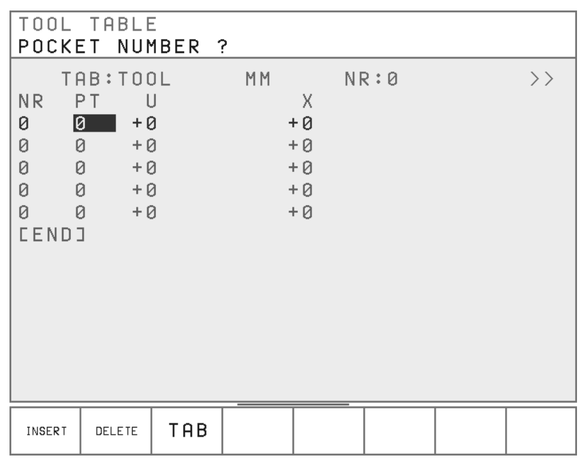
You can define and store up to 999 tools and their tool data in a tool table. You can assign a pocket number in the tool magazine to the tools.

With MP7261 you can limit the number of pockets in the tool magazine. There is no limiting if MP7261=0. Setting MP7265=1 prevents multiple assignment of a pocket number.

Tool table: Standard tool data

Abbr.	Input	Dialog
NR	Number by which the tool is called in the program (e.g. 5)	**Tool number ?**
PT	Pocket number in the tool magazine	**Pocket number?**
U	Tool undersize (diametrical)	**Tool undersize? (diametrical)**
X	Tool compensation value for the X axis	**Tool compensation?**
Y	Tool compensation value for the Y axis	**Tool compensation?**
Z	Tool compensation value for the Z axis	**Tool compensation?**
C	Tool compensation value for the C axis	**Tool compensation?**
R	Compensation value for the tool radius R	**Tool radius R?**

Tool undersize from the table is only active if you do not define it again during TOOL CALL.



Editing tool tables

The tool table that is active during execution of the part program is designated TOOL.T. It can only be edited in the Programming and Editing mode. Other tool tables that are used for archiving or test runs are given different file names with the extension .T .

To open any other tool table:

- Select the Programming and Editing mode of operation.

PGM NAME

- Call the program directory.
- Choose the desired TOOL table, and confirm your choice with the ENT key or with the SELECT soft key.

When you have opened the tool table, you can edit the tool data by moving the cursor to the desired position in the table with the arrow keys or the soft keys. You can overwrite the stored values, or enter new values at any position. The available editing functions are illustrated in the table below.

If the TNC cannot show all positions in the tool table in one screen page, the highlight bar at the top of the table will display the symbol "**>>**" or "**<<**".

Exiting the tool table

- Call the file manager and select a file of a different type, e.g. a part program.

Editing functions for tool tables	Soft key
Insert new line above the highlighted field	INSERT
Delete line	DELETE
Create new TOOL table by entering a new name	TAB

Calling electrode data

Electrode data are called into the part program with TOOL CALL. TOOL CALL is programmed with

- Tool number
- Spindle axis
- Undersize
- Code indicating whether the electrode is a following electrode

You can skip individual entries with NO ENT, for example to enter only one (new) undersize.

Calling electrode data

TOOL CALL

- Select the tool call function with the TOOL CALL key.
- **`Tool number:`** Enter the number of the electrode as defined in TOOL DEF block. Confirm your entry with the ENT key.
- **`Working tool axis X/Y/Z/4:`** Enter the tool axis, e.g. Z.
- **`Tool undersize (diameter):`** Enter the electrode undersize (diameter), e.g. 0.5. Confirm with the ENT key or
  skip the entry with the NOENT key.
- **`Folw. electrode YES=ENT/NO=NOENT:`** e.g., to identify the electrode as a following electrode.

If you define a value for the tool undersize in the TOOL CALL, the value from the TOOL table is ignored. Otherwise the undersize from the TOOL table is valid.

Example: Electrode call

Call electrode number 5 in the tool axis Z.
The diametrical electrode undersize is + 0.5 mm.

```
20 TOOL CALL 5 Z U+0.5 F
```

Following electrode

Answering "YES" to FOLW. ELECTRODE prevents the workpiece from being damaged by too large an amount of taper (caused by insufficient flushing or deep mold cavities) during roughing operations at high current. For the gap between the electrode and the workpiece the TNC multiplies the minimum gap by the value in Q157.
The value in Q157 is determined by your answer to FOLW. ELECTRODE.

Call with following electrode: finishing, small undersize (narrow gap): Q157 = 1

Call without following electrode: roughing, large undersize (wide gap): 1< Q157 < 2.5

Changing the electrode

The electrode can be changed automatically or manually.

Automatic electrode change with EL-CALL

The function for automatic electrode change is adapted to the TNC by the machine tool builder. Refer to your machine tool manual.

If your machine features an automatic handling system, you can program an automatic electrode change with the EL-CALL function. EL-CALL combines the functions TOOL DEF and TOOL CALL.

- Select the Programming and Editing mode of operation.

EL-CALL

- Select the **`EL-CALL function`** with the EL-CALL soft key.
- **`Electrode name:`** Enter the name of the electrode, e.g. 1. You can enter up to 16 characters (letters and numbers). Confirm your entry with the ENT key.
- **`Working tool axis X/Y/Z/4:`** Enter the tool axis.
- **`Folw. electrode YES=ENT/NO=NOENT:`** e.g., to identify the electrode as a following electrode.

Example

```
4 EL-CALL 1    /ZF
```

To ensure that the TNC shifts the electrode correctly to the programmed position, you must enter the correct C axis coordinates in a traversing block with L, programmed directly after the EL-CALL block.

Manual electrode change

Before a manual electrode change, you must move the electrode to a changing position. Course of actions:

- Interrupt program run (see "Interrupting machining" on page 223)
- Move the electrode to the change position (can be programmed)
- Change electrode
- Resume the program run (see "Resuming program run after an interruption" on page 225)

Electrode change position

The electrode change position must be capable of being approached without collision next to or over the workpiece.
The coordinates of the change position can also be entered as machine-based coordinates with miscellaneous functions M91 and M92.
If TOOL CALL 0 is programmed before the first electrode call, the TNC moves the clamping shaft in the spindle axis to a position that is independent of the electrode length.

Electrode compensation

You can compensate the electrode length and radius in a separate program block.

- Select the Programming and Editing mode of operation.

EL-CORR

- Select the **EL-CORR function** with the EL-CORR soft key.
- **Undersize comp:** Enter the undersize compensation. Confirm your entry with the ENT key.
- **Electrode length comp:**
  Enter the electrode length compensation value.
  Confirm your entry with the ENT key.
  If no electrode length compensation: Press the NO ENT key.
- **Electrode radius comp:**
  Enter the electrode radius compensation value.
  Confirm your entry with the ENT key.
  If no electrode radius compensation: Press the NO ENT key.

Example

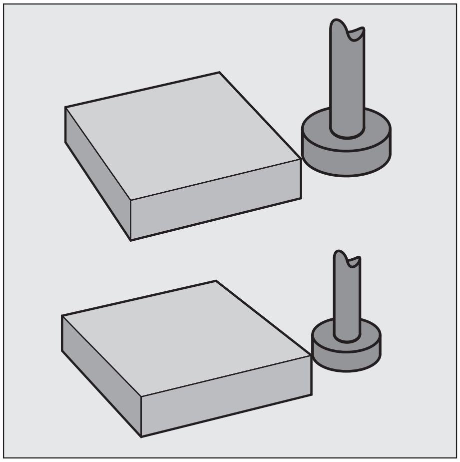
```
4 EL-CORR U+1 L R+0.5
```

Effect on Q parameters

The EL CORR block influences the pre-assigned Q parameters Q108, Q158 and Q159 (see also "Electrode data: Q108, Q158 to Q160" on page 203).

5.2 Electrode Compensation Values

For each electrode, the TNC takes the compensation value for the electrode length into account for the electrode axis. In the working plane, it compensates the electrode radius.



Electrode length compensation

The compensation value for the electrode length goes into effect automatically as soon as an electrode is called and the spindle axis is moved.
The compensation value for the electrode length is cancelled by calling an electrode with length L = 0.

> ☞ If a positive length compensation was active before TOOL CALL 0, the distance to the workpiece will be reduced.
> If the electrode axis is positioned incrementally immediately following a TOOL CALL, then in addition to the programmed value the length difference between the old and new electrodes will also be traversed.

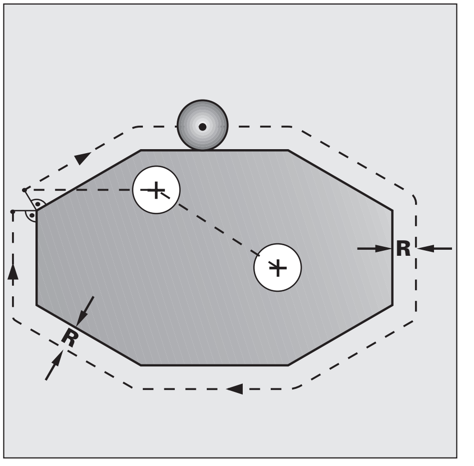
Electrode radius compensation

Radius compensation becomes effective as soon as an electrode is called and is moved in the working plane with RL or RR.
To cancel radius compensation, program a positioning block with R0.

Electrode movements can be programmed in the following ways:

- Without radius compensation: R0
- With radius compensation: RL or RR
- Paraxial movements with R+ or R–

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

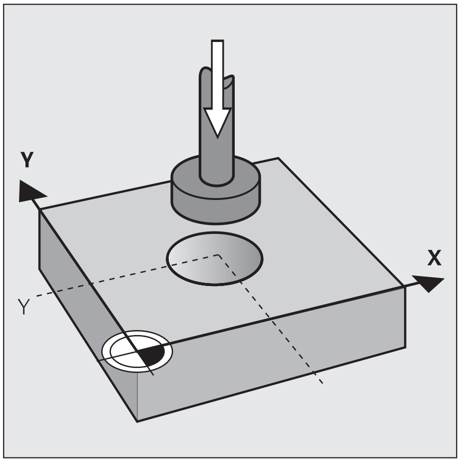


Contouring without radius compensation: R0

The electrode center moves to the programmed coordinates.

Applications:

- Countersinking
- Pre-positioning

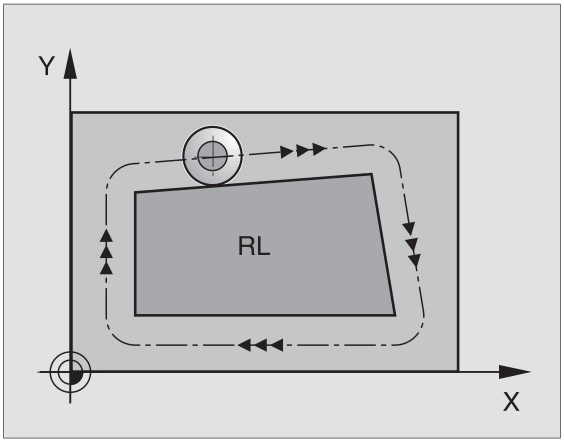


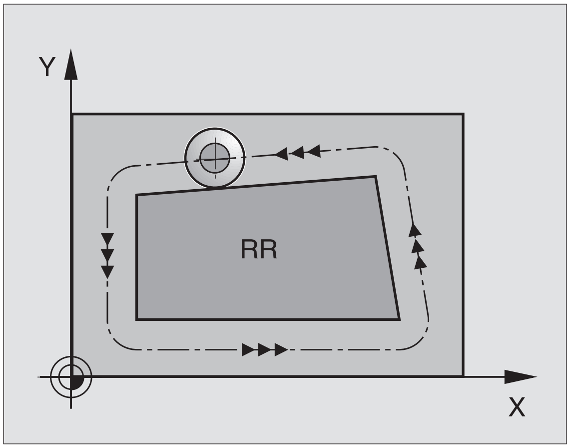
Tool movements with radius compensation: RR and RL

RR The electrode moves to the right of the programmed contour.

RL The electrode moves to the left of the programmed contour.

The electrode center moves along the contour at a distance equal to the radius. "Right" or "left" are to be understood as based on the direction of electrode movement along the workpiece contour. See figures at right.





Between two program blocks with different radius compensations (**RR** and **RL**) you must program at least one traversing block in the working plane without radius compensation (that is, with **R0**).

Radius compensation does not take effect until the end of the block in which it is first programmed.

Whenever radius compensation is activated or cancelled, the TNC positions the electrode perpendicular to the programmed starting or end position. Position the electrode at a sufficient distance from the first (or last) contour point to prevent damaging the contour.

Entering radius compensation

Program any desired path function, enter the coordinates of the target point and confirm your entry with ENT.

RADIUS COMP. RL/RR/NO COMP. ?

To select tool movement to the left of the contour, press the RL key, or

To select tool movement to the right of the contour, press the RR key, or

ENT To select tool movement without radius compensation or to cancel radius compensation, press the ENT key.

END To terminate the block, press the END key.

Shortening or lengthening paraxial movements: R+, R–

This type of radius compensation is only possible for single-axis movements in the working plane. The programmed electrode path is lengthened (R–) or shortened (R+).

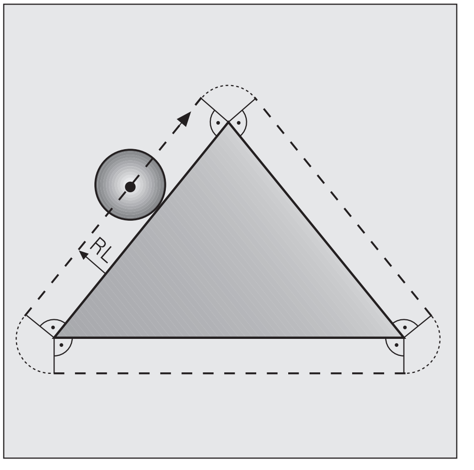
Applications:

- Paraxial machining
- Under certain circumstances for pre-positioning the electrode

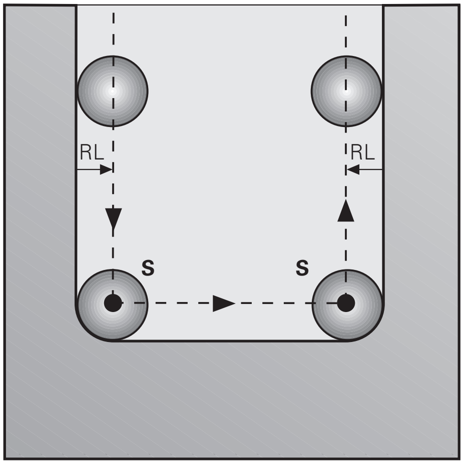
> R+ and R– are available when a positioning block is opened with an orange axis key.

Radius compensation: Machining corners

- Outside corners
  The TNC moves the electrode in a transitional arc around outside corners. The electrode "rolls" around the corner point. If necessary, the feed rate F of the electrode is automatically reduced at outside corners to reduce machine stress, for example at very great changes of direction.



- Inside corners
  The TNC calculates the intersection of the electrode center paths at inside corners. It then starts the next contour element from this point. This prevents damage to the workpiece at the inside corners. The permissible electrode radius is therefore limited by the geometry of the programmed contour.



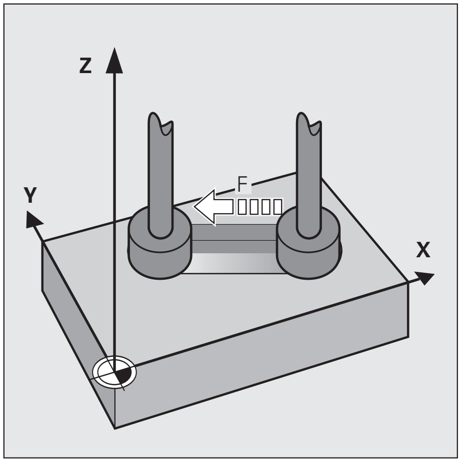
> To prevent the tool from damaging the contour, be careful not to program the starting or end position for machining inside corners at a corner of the contour.

5.3 Entering Electrode-Related Data

Introduction

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

- Feed rate F
- Miscellaneous functions M



Feed rate F

The feed rate is the speed (in millimeters per minute or inches per minute) at which the electrode center moves. For eroding, the feed rate is defined by machine parameters. It can also be selected for positioning with eroding.

Input range:

F = 0 to 30 000 mm/min (1 181 inch/min)

The maximum feed rates can be different for the individual axes and are set in machine parameters.

Input

Answer the dialog question in the positioning block:

FEED RATE F=? / F MAX = ENT

100 ENT Enter the feed rate, for example F = 100 mm/min.

The TNC does not always ask for FMAX.

Rapid traverse

For rapid traverse you can enter F = FMAX. The rapid traverse can also be programmed directly.
FMAX is only effective in the program block in which it is programmed.

Duration of feed rate F

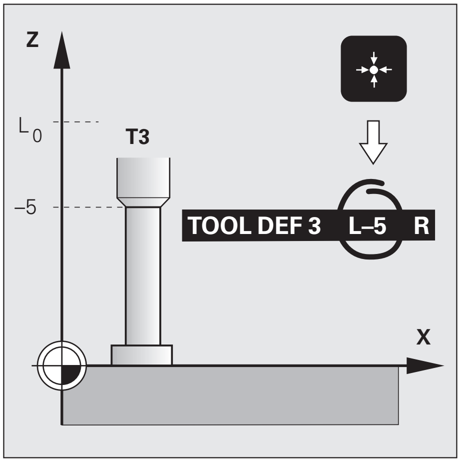
A feed rate entered as a numerical value remains in effect until the control encounters a block with a different feed rate. If the new feed rate is FMAX, then after the block with FMAX is executed, the feed rate will return to the last feed rate entered as a numerical value.

Feed rate override

You can adjust the feed rate with the override knob on the TNC keyboard.

5.4 Actual Position Capture

Function



The coordinates of the electrode position can be transferred into the part program with the actual position capture feature.

You can also use this feature to transfer the electrode length directly into the program (also see "To enter the electrode data into a program block" on page 73).

When the ACTL, ACT.W, NOML, NOM.W or REF positions are being displayed, the TNC takes the value from the position display. When the DIST. or LAG positions are being displayed, the TNC uses the associated nominal value.

Actual position capture

Select the MANUAL OPERATION mode.

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

Select the PROGRAMMING AND EDITING mode of operation.

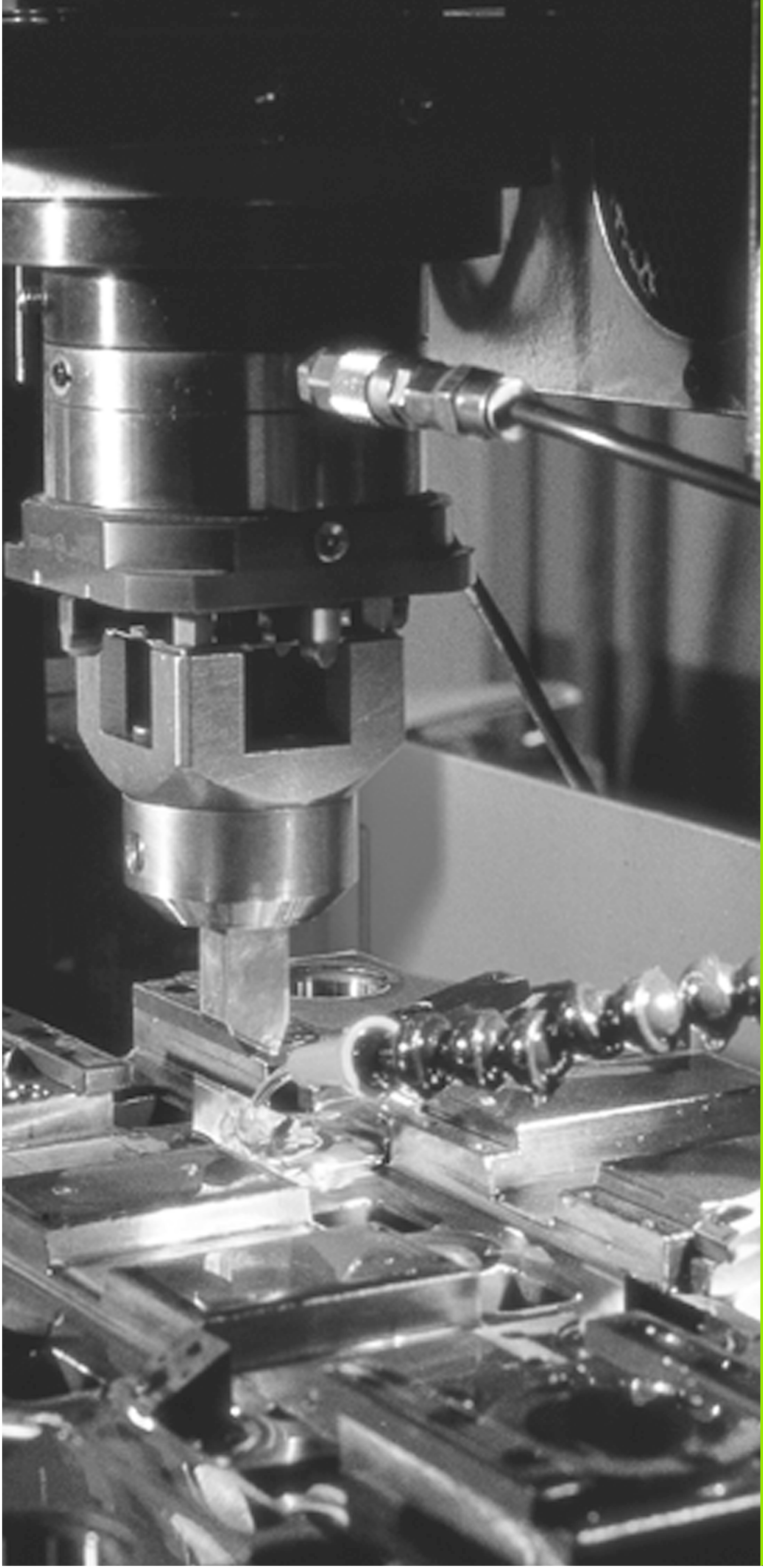
Select or open the program block into which you wish to transfer the actual position of the electrode.

COORDINATES ?

X Select the axis in which you wish to capture a coordinate, such as X.

Transfer the actual position coordinate into the program.

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



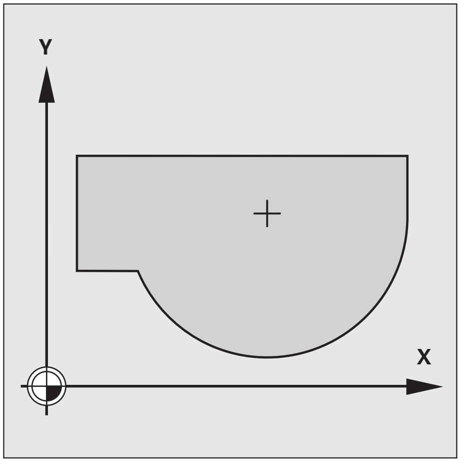
6

Programming: Programming Contours

6.1 General Information on Programming Electrode Movements

Electrode movements are always programmed as if the electrode moves and the workpiece remains stationary.

Before running a part program, always pre-position the electrode to prevent the possibility of damaging it or the workpiece.



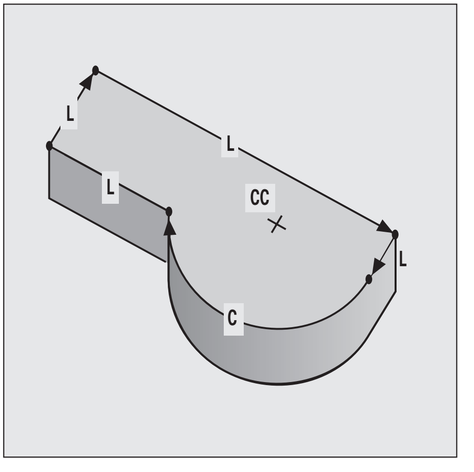
Path functions

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 elements (helical paths):

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



Machines with 5 axes

A fifth axis can only be moved in the operating modes MANUAL OPERATION or ELECTRONIC HANDWHEEL or with a "PLC positioning."

Contact your machine tool builder if you need to position a fifth axis.

Subprograms and program section repeats

If a machining routine 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:

- Repeat a machining routine immediately after it is executed (program section repeat)
- Writing a machine routine separately and then inserting it into a program (subprogram)
- Calling a separate program for execution or test run within the main program (program call)

Cycles

The ORBIT erosion cycle is the basis for user-specific machining tasks. This cycle allows you to program features such as conical and rounded cavities.
You can also define the eroding time for this cycle.

Further cycles for coordinate transformations are available. These can be used to change the coordinates of a machining sequence in a defined way. Examples:

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

The TOOL DEF cycle allows you to enter compensation values for the electrode dimensions (tool data).

Parametric programming

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

- Conditional and unconditional jumps
- Probing for measurements with an electrode during program run
- Output of values and messages
- Transferring values to and from memory

The following mathematical functions are available:

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

6.2 Contour Approach and Departure

A convenient way to approach or depart the workpiece is on an arc which is tangential to the contour. This is done with the corner rounding function (see "Corner rounding RND" on page 97).

Starting point and end point of machining

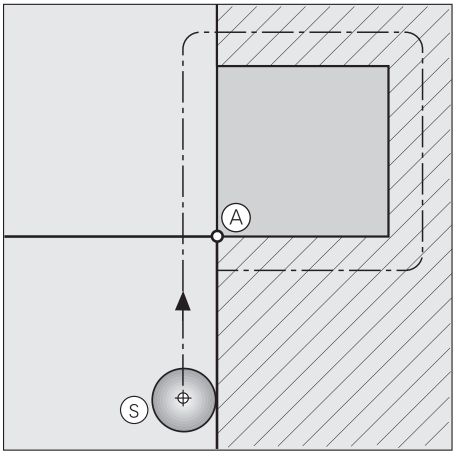
Starting point S

From the starting point S the electrode approaches the first contour point A.
The starting point is programmed without radius compensation.

The starting point S must be:

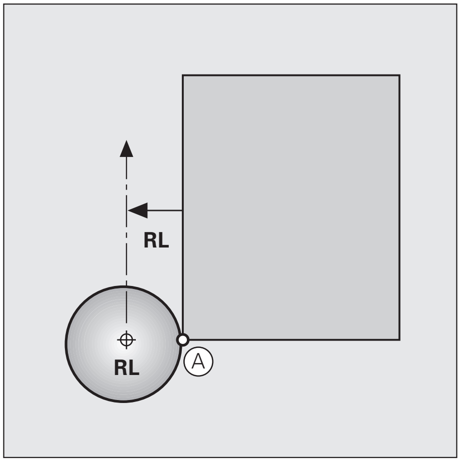
- Approachable without danger of collision
- Close to the first contour point
- Located in relation to the workpiece such that no contour damage can occur when the contour is approached.

If the starting point S is located within the hatched area, the contour will be damaged when the first contour point is approached.
The ideal starting point is located on the extended tool path for machining the first contour element.



First contour point A

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



Approaching the starting point S in the spindle axis

When the starting point **S** 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.

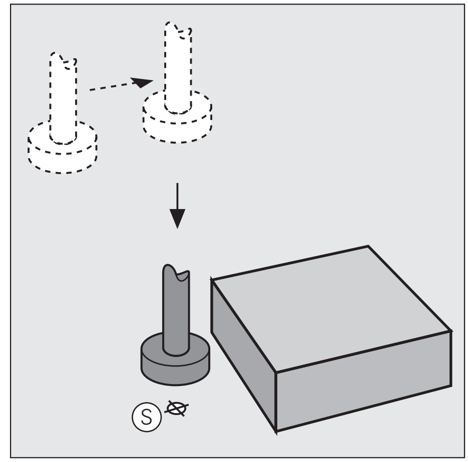
Example:

```
L X ... Y ...
```

The electrode retains the Z coordinate and moves in the XY plane to the start position.

```
L Z-10
```

The electrode is positioned in the Z axis to working depth.

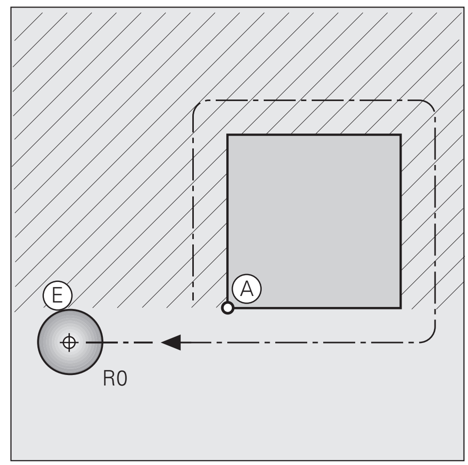


End point

Similar requirements hold for the end point E:

- Approachable without danger of collision
- Near the last contour point
- Avoids damage to tool and the workpiece

The ideal location for the end point E is again on the extended tool path outside the hatched area. It is approached without radius compensation.



Departure from an end point in the spindle axis

The spindle axis is moved separately.

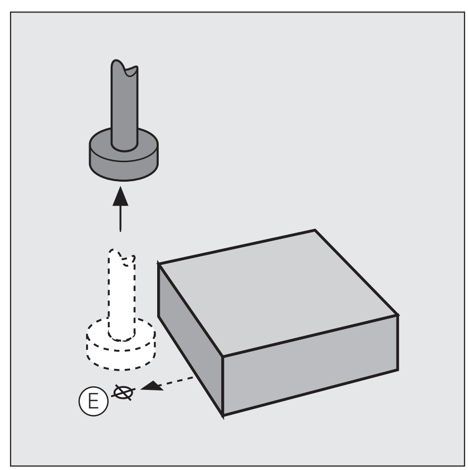
Example:

```
L X ... Y ... R0
```

The electrode retains the Z coordinate and moves in the XY plane to the end position.

```
L Z+50
```

The electrode moves to set-up clearance.

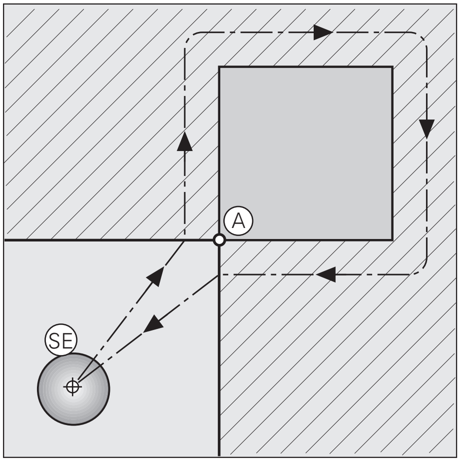


Common starting and end point

Outside of the hatched area in the illustration it is possible to define a single point as both the starting and end point.

The ideal location for this 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.



Tangential contour approach and departure

Starting point and end point

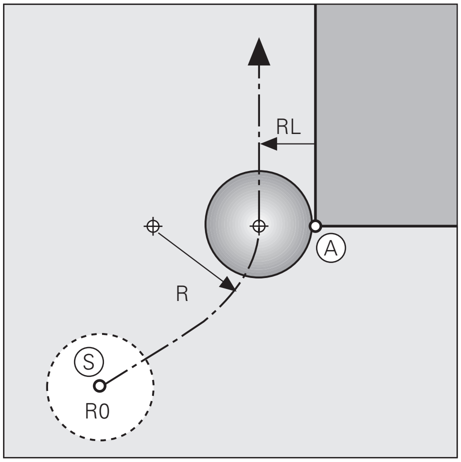
The starting point and end point of machining are off the workpiece near the first or last contour element.

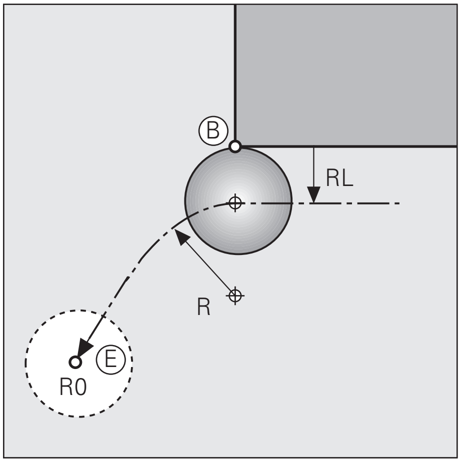
The tool path to the starting or end point is programmed without radius compensation.

Input

The RND function is entered at the following points in the program:

- For the approach path, RND is programmed after the block containing the first contour element (the first block with radius compensation RL/RR).
- For the departure path, RND is programmed after the block containing the last contour element (the last block with radius compensation RL/RR).





Example NC blocks

`...`	
`7 L X ... Y ... R0`	Starting point S
`8 L X ... Y ... RL`	First contour point A
`9 RND R`	Tangential approach
`...`	
`CONTOUR ELEMENTS`	
`...`	
`52 L X ... Y ... RL`	Last contour point B
`53 RND R`	Tangential departure
`54 L X ... Y ... R0`	End point E
`...`	

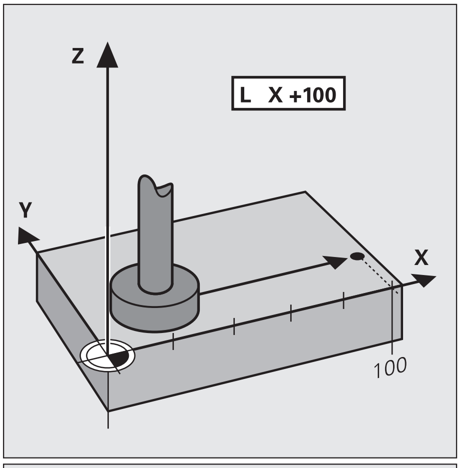
The radius in the RND function must be selected such that it is possible to perform the circular arc between the contour point and the starting point or end point.

6.3 Path functions

General

Part program input

You create a part program by entering the workpiece dimensions. Coordinates are programmed as absolute or relative (incremental) values.
In general you program the coordinates of the end point of the contour element.
The TNC automatically calculates the path of the electrode based on the electrode data (length and radius) and the radius compensation.



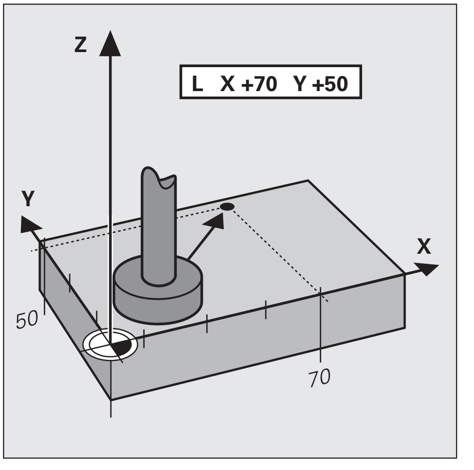
Programmed machine axis movement

All axes programmed in a single NC block are moved simultaneously.

Paraxial movements

The electrode moves in path parallel to the programmed axis.

Number of axes programmed in the NC block: 1



Movement in the main planes

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

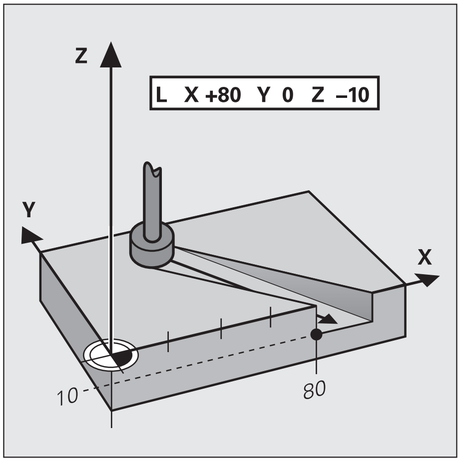
Number of axes programmed in the NC block: 2

Movement of three machine axes (3-D movement)

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

Number of axes programmed in the NC block: 3

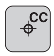
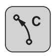
Exception:
A helical path is created by combining circular movement with linear movement.



6.4 Path Contours — Cartesian Coordinates

Overview of path functions

The path function keys define the type of contour element and open a programming dialog.

Function	Path function key	Tool movement	Required input
Line **L**		Straight line	Coordinates of the end points of the straight line
Circle Center **CC**		No tool movement	Coordinates of the circle center or pole
Circle **C**		Circular arc around a circle center CC to an arc end point	Coordinates of the arc end point, direction of rotation
Circular Arc **CR**		Circular arc with a certain radius	Coordinates of the arc end point, arc radius, direction of rotation
Circular Arc **CT**		Circular arc with tangential connection to the preceding and subsequent contour elements	Coordinates of the arc end point
Corner Rounding **RND**		Circular arc with tangential connection to the preceding and subsequent contour elements	Rounding-off radius R

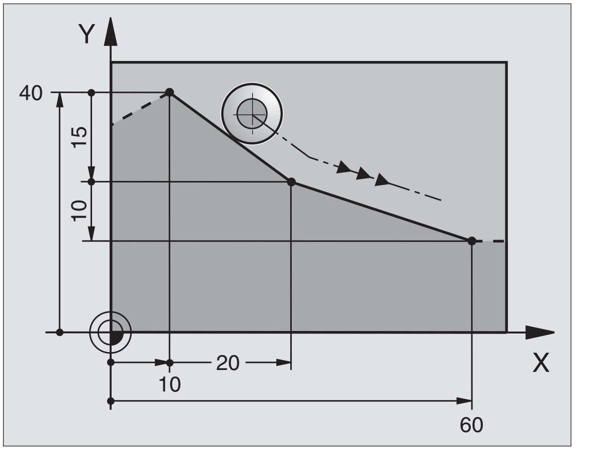
Straight line L

The TNC moves the tool in a straight line from its current position to the straight-line end point. The starting point is the end point of the preceding block.

- **Coordinates** of the end point of the straight line

Further entries, if necessary:

- **Radius compensation RL/RR/R0**
- **Feed rate F**
- **Miscellaneous function M**



Example NC blocks

```
7 L X+10 Y+40 RL F M
8 L IX+20 IY-15 R F M
9 L X+60 IY-10 R F M
```

Actual position capture

You can also generate a straight-line block (L block) by using the ACTUAL-POSITION-CAPTURE key:

- In the Manual Operation mode, move the tool to the position you wish to capture.
- Switch the screen display to Programming and Editing.
- Select the program block after which you want to insert the L block.

- Press the ACTUAL-POSITION-CAPTURE key: The TNC generates an L block with the actual position coordinates.

In the MOD function, you define the number of axes that the TNC saves in an L block.

Programming a straight line

Example — programming a straight line:

Initiate the programming dialog, e.g. for a straight line.

```
COORDINATES ?
```

X **10**

Y **5**

ENT

Enter the coordinates of the straight-line end point.

```
RADIUS COMP. RL/RR/NO COMP. ?
```

Select the radius compensation (here, press the RL soft key - the tool moves to the left of the programmed contour).

ENT

Move the electrode on the straight line directly to the end point

```
FEED RATE F=? / F MAX = ENT
```

100 ENT

Enter the feed rate (here, 100 mm/min), and confirm your entry with ENT.

ENT

Choose rapid traverse for the electrode:
F = F MAX

```
MISCELLANEOUS FUNCTION M ?
```

37 ENT

Enter a miscellaneous function, for example M37 (Eroding OFF).

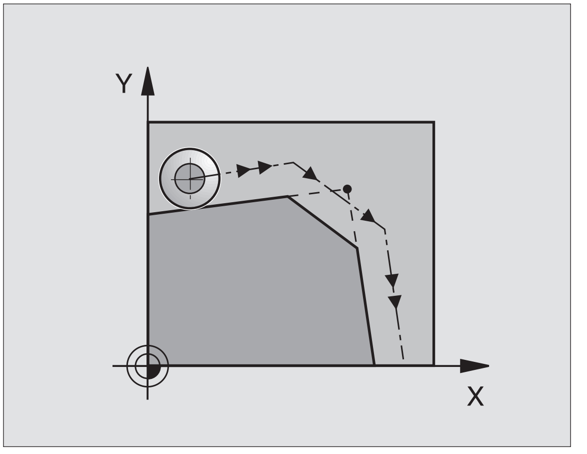
The part program now contains the following line:

```
L X+10 Y+5 RL F MAX M37
```

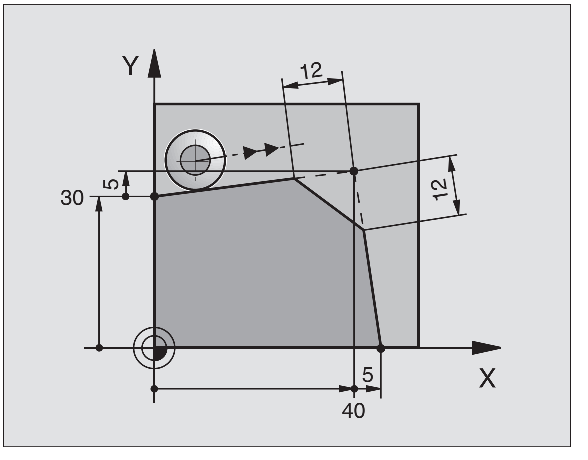
Inserting a chamfer CHF between two straight lines

The chamfer enables you to cut off corners at the intersection of two straight lines.

- The blocks before and after the CHF block must be in the same working plane.
- 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.
  (The tool radius in the illustration at bottom right is too large)



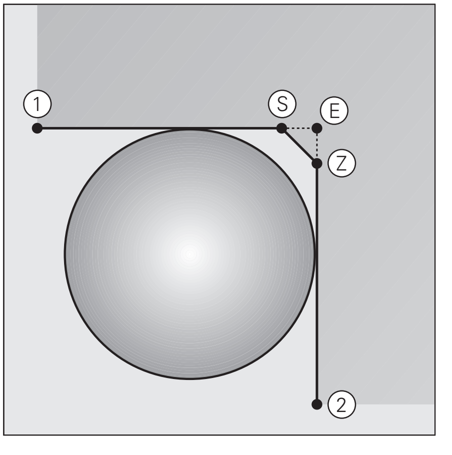
- **Chamfer side length:** Input the length L without entering an axis designation



Example NC blocks

```
7 L X+0 Y+30 RL F M
8 L X+40 IY+5 R F M
9 L 12
10 L IX+5 Y+0 R F M
```

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

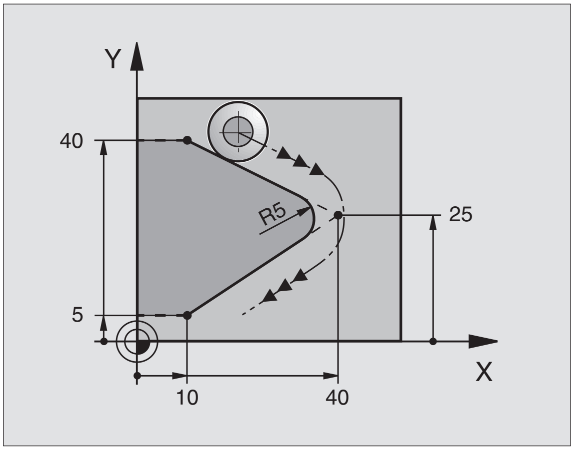


Corner rounding RND

The RND function is used for rounding off corners.

The tool moves on an arc that is tangentially connected to both the preceding and subsequent contour elements.

The rounding arc must be large enough to accommodate the tool.



- **Rounding-off radius**: Enter the radius

Further entries, if necessary:
- **Feed rate F** (only effective in RND block)

Example NC blocks

```
5 L X+10 Y+40 RL F M
6 L X+40 Y+25 R F M
7 RND R5 R F M
8 L X+10 Y+5
```

In the preceding and subsequent contour elements, both coordinates must lie in the plane of the rounding arc. If you machine the contour without tool-radius compensation, you must program both coordinates in the working plane.

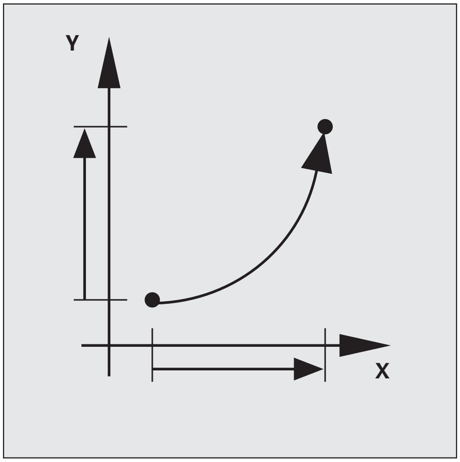
The corner point is cut off by the rounding arc and is not part of the contour.

A feed rate programmed in the RND block is effective only in that block. After the RND block, the previous feed rate becomes effective again.

You can also use an RND block for a tangential contour approach.

Circles and circular arcs

Here the TNC moves two machine axes in a circular path relative to the workpiece. The axes can also be auxiliary axes U, V, or W.

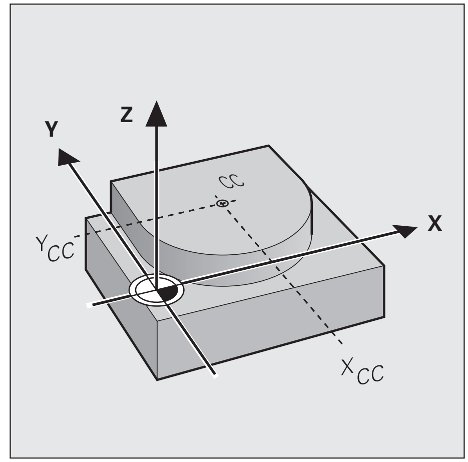


Circle center CC

You can define a circle center CC for circles that are programmed with the C key (circular path C). This is done in the following ways:

- Entering the Cartesian coordinates of the circle center, or
- Using the circle center defined in an earlier block, or
- Capturing the coordinates with the ACTUAL-POSITION-CAPTURE key.

- **`Coordinates`** CC: Enter the circle center coordinates, or
  If you want to use the last programmed position, do not enter any coordinates.



Example NC blocks

```
5 CC X+25 Y+25
```

or

```
10 L X+25 Y+25
11 CC
```

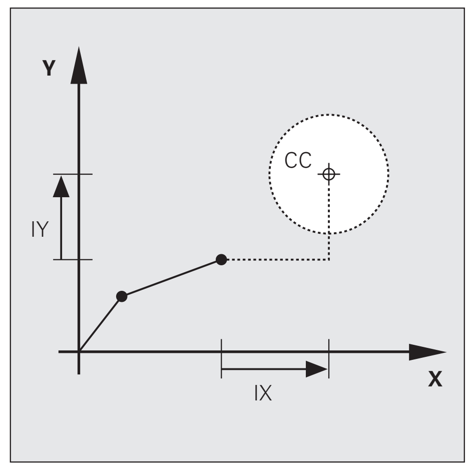
The program blocks 10 and 11 do not refer to the illustration.

Duration of effect

The circle center definition remains in effect until a new circle center is programmed. You can also define a circle center for the secondary axes U, V and W.

Entering the circle center CC incrementally

If you enter the circle center with incremental coordinates, you have programmed it relative to the last programmed position of the tool.

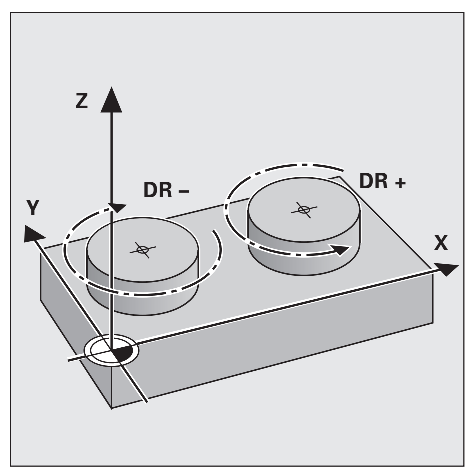


- The only effect of CC is to define a position as circle center: The tool does not move to this position.
- The circle center is also the pole for polar coordinates.

Direction of rotation DR

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

- Clockwise rotation: negative direction of rotation (DR–)
- Counterclockwise rotation: positive direction of rotation (DR+)



Radius compensation in circular paths

You cannot begin radius compensation in a circle block - it must be activated beforehand in a line block (L block).

Circles in the main planes

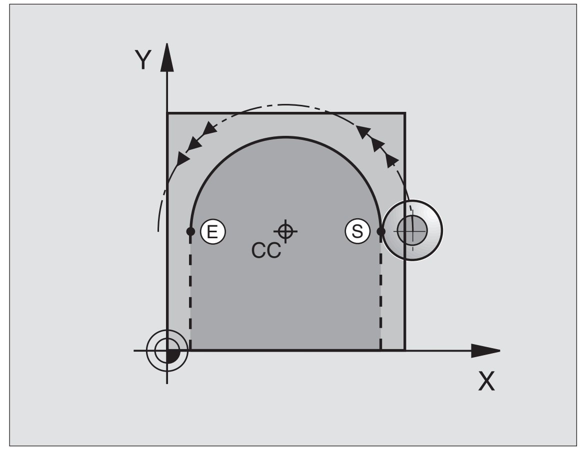
When you program a circle, the TNC assigns it to one of the main planes. This plane is automatically defined when you set the electrode axis during an electrode call (TOOL CALL).

Tool axis	Main plane
Z	**XY**, also UV, XV, UY
Y	**ZX**, also WU, ZU, WX
X	**YZ**, also VW, YW, VZ

You can program circles that do not lie parallel to a main plane by using Q parameters.

Circular path C around circle center CC

Before programming a circular path C, you must first enter the circle center CC. The last programmed tool position before the C block is used as the circle starting point.



> If you are using an electrode with tool compensation in the XY plane, you must rotate the electrode in synchrony with the angle on circular arcs. For example, for a semicircle you must rotate the C axis by 180° (incremental).

- Move the tool to the circle starting point.
  - **Coordinates** of the circle center
  - **Enter the coordinates of the arc end point**
  - **Direction of rotation DR**

  Further entries, if necessary:
  - **Linear coordinates**
  - **Feed rate F**
  - **Miscellaneous function M**

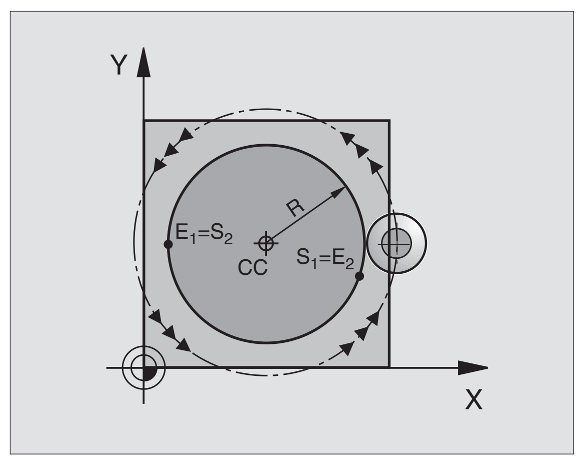
Example NC blocks

```
5 CC X+25 Y+25
6 L X+45 Y+25 RR F M
7 C X+5 Y+25 IC +180 DR+ R F M
```

Full circle

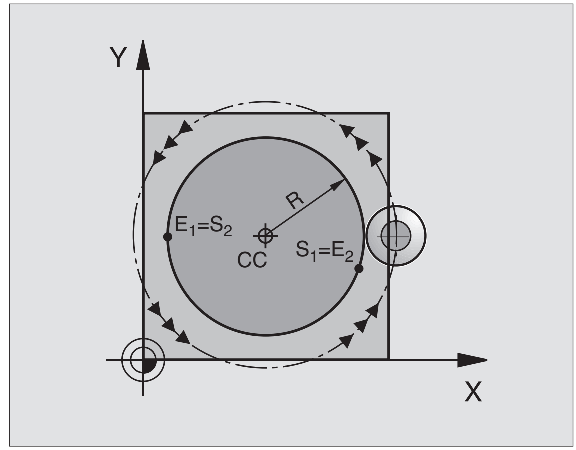
To program a full circle you must enter two C blocks in succession: The end point of the first semicircle is the starting point of the second circle. The end point of the second semicircle is the starting point of the first.

The easiest method of programming a full circle is described on page 111.



Circular path CR with defined radius

The electrode moves on a circular path with the radius R.



- **Coordinates** of the arc end point
- **Radius R**
  Note: The algebraic sign determines the size of the arc!
- **Direction of rotation DR**
  Note: The algebraic sign determines whether the arc is concave or convex!

Further entries, if necessary:

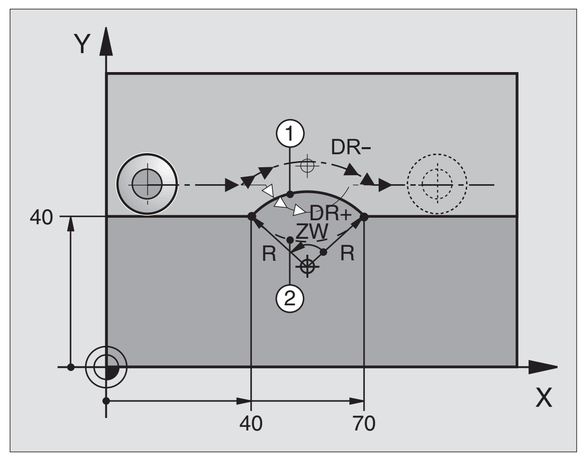
- **Miscellaneous function M**
- **Feed rate F**

Full circle

To program a full circle you must enter two CR blocks in succession: The end point of the first semicircle is the starting point of the second circle. The end point of the second semicircle is the starting point of the first.

The easiest method of programming a full circle is described on page 111.

Central angle CCA and arc radius R



The starting and end points on the contour can be connected with four arcs of the same radius:

Smaller arc: CCA<180°
Enter the radius with a positive sign R>0

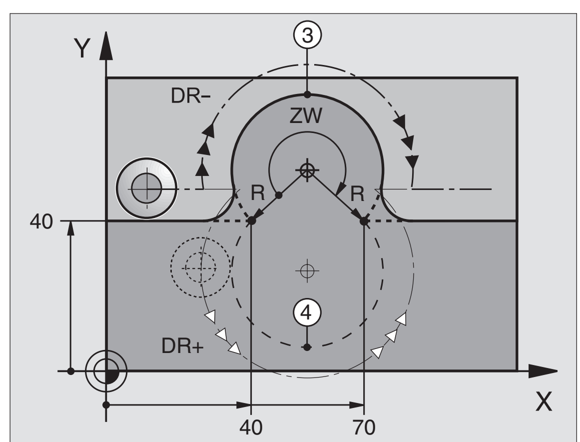
Larger arc: CCA>180°
Enter the radius with a negative sign R<0

The direction of rotation determines whether the arc is curving outward (convex) or curving inward (concave):

Convex: Direction of rotation DR– (with radius compensation RL)

Concave: Direction of rotation DR+ (with radius compensation RL)

Example NC blocks




```
10 L X+40 Y+40 RL F M36
11 CR X+70 Y+40 R+20 DR- (ARC 1)
```

or

```
11 CR X+70 Y+40 R+20 DR+ (ARC 2)
```

or

```
11 CR X+70 Y+40 R-20 DR- (ARC 3)
```

or

```
11 CR X+70 Y+40 R-20 DR+ (ARC 4)
```

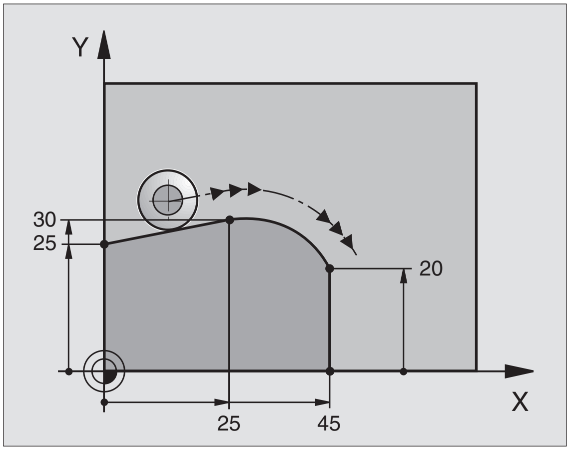
- For a full circle, two CR blocks must be programmed in succession.
- The distance from the starting and end points of the arc diameter cannot be greater than the diameter of the arc.
- The maximum radius is 30 m.
- You may **not** enter rotary axes A, B or C.

Circular path CT with tangential connection

The electrode moves on an arc that starts at a tangent with the previously programmed contour element.

A transition between two contour elements is called tangential when there is no kink or corner at the intersection between the two contours—the transition is smooth.

The contour element to which the tangential arc connects must be programmed immediately before the CT block. This requires at least two positioning blocks.



If you are using an electrode with tool compensation in the XY plane, you must rotate the electrode in synchrony with the angle on circular arcs. For example, for a semicircle you must rotate the C axis by 180° (incremental).

- **Coordinates** of the arc end point

Further entries, if necessary:

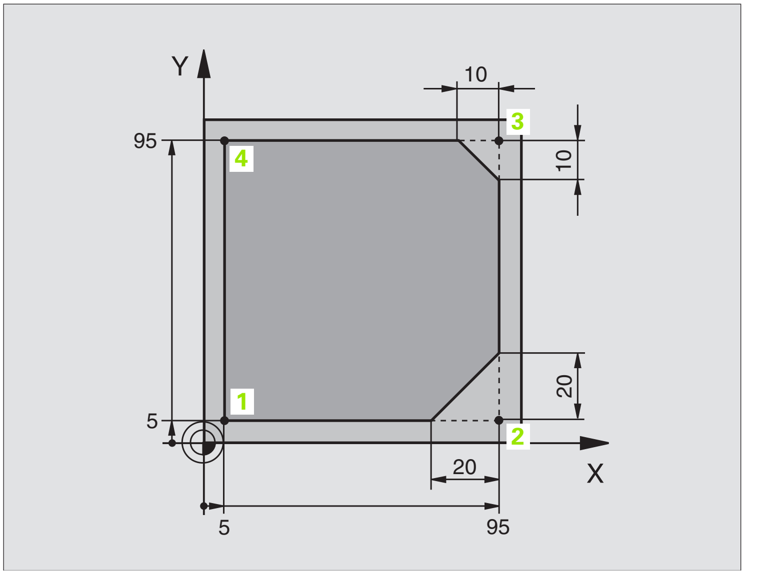
- **Feed rate F**
- **Miscellaneous function M**

Example NC blocks

```
7 L X+0 Y+25 RL F M36
8 L X+25 Y+30 R F M
9 CT X+45 Y+20 R F M
10 L Y+0 R F M
```

A tangential arc is a two-dimensional operation: the coordinates in the CT block and in the contour element preceding it must be in the same plane of the arc.

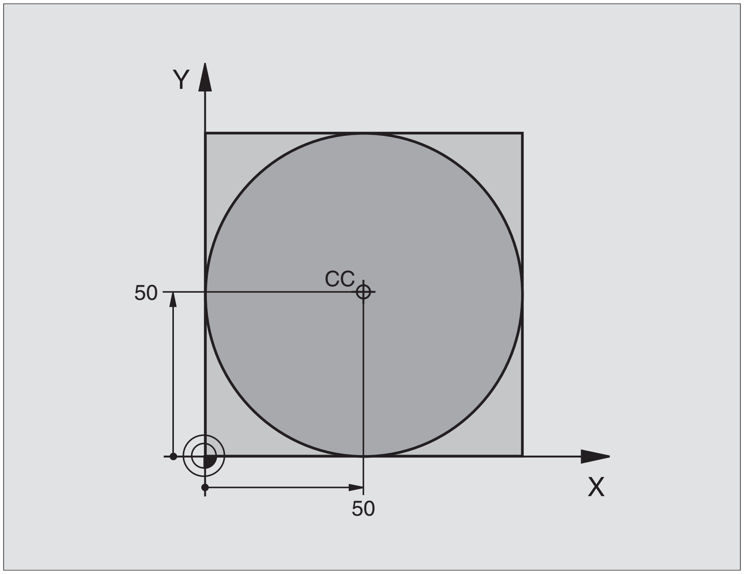
Example: Linear movements and chamfers with Cartesian coordinates



`0  BEGIN PGM LINEAR MM`	Start of program, program name LINEAR, dimensions in mm
`1  BLK FORM 0.1 Z X+0 Y+0 Z-20`	Define blank form for graphic workpiece simulation
`2  BLK FORM 0.2 X+100 Y+100 Z+0`	
`3  CYCL DEF 1.0 GENERATOR`	Cycle GENERATOR (see "Cycle 1 GENERATOR" on page 133)
`4  CYCL DEF 1.1 P-TAB CUST`	Select erosion table (here, table CUST)
`5  CYCL DEF 1.2 MAX=3 MIN=3`	Set power stage (here, to stage 3)
`6  TOOL DEF 1 L+0 R+15`	Define electrode in the program
`7  TOOL CALL 1 Z U+1`	Call electrode in the infeed axis Z, undersize 1 mm
`8  L Z+100 C+0 R0 F MAX M`	Retract in the infeed axis; orient electrode; rapid traverse
`9  L X-10 Y-10 R F MAX M`	Pre-position in X and Y; rapid traverse
`10  L Z–10 R F MAX M`	Move to working depth
`11  LX+5 Y+5 RL F M36`	Approach the contour at point 1 with radius compensation; eroding ON
`13  L Y+95 R F M`	Move to point 2
`14  L X+95 R F M`	Point 3: first straight line for corner 3
`15  L 10`	Program chamfer with length 10 mm

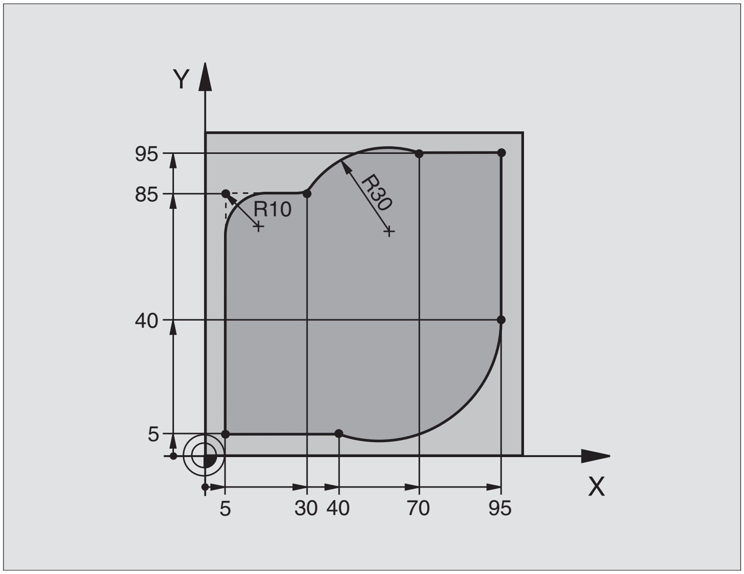
`16  L Y+5 R F M`	Point 4: 2nd straight line for corner 3, 1st straight line for corner 4
`17  L 20`	Program chamfer with length 20 mm
`18  L X+5 R F M`	Move to last contour point 1, second straight line for corner 4
`21  L X–10 Y–10 R0 F M37`	Retract tool in the working plane; eroding OFF
`20  L Z+100 F MAX`	Move electrode to set-up clearance; rapid traverse
`21 END PGM LINEAR MM`	End of program

Example: Full circle with Cartesian coordinates



0 BEGIN PGM C-CC MM	Start of program
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Define the workpiece blank
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 CYCL DEF 1.0 GENERATOR	Cycle GENERATOR (see "Cycle 1 GENERATOR" on page 133)
4 CYCL DEF 1.1 P-TAB HDH700	Select erosion table (here, table HDH700)
5 CYCL DEF 1.2 MAX=6 MIN=6	Set power stage (here, to stage 6)
6 TOOL DEF 6 L+0 R+15	Define electrode in the program
7 TOOL CALL 6 Z U+1.5	Call electrode in the infeed axis Z, undersize 1.5 mm
8 L Z+250 C+0 R0 F MAX M37	Set-up clearance; orient electrode; eroding OFF
9 CC X+50 Y+50	Define the circle center
10 L X-40 Y+50 R0 F MAX M	Pre-position the tool
11 L Z-5 R0 F MAX M	Move to working depth
12 L X+0 Y+50 RL M36	Move to first contour point with radius compensation; eroding ON
13 C X+100 Y+50 DR- R F M	End point of first semicircle; clockwise rotation
14 C X+0 Y+50 DR- R F M	End point of second semicircle; clockwise rotation
15 X-40 Y+50 R0 F MAX M37	Retract tool in the working plane; eroding OFF
16 L Z+250 F MAX	Move electrode to set-up clearance; rapid traverse
17 END PGM C-CC MM	End of program

Example: Circular movements with Cartesian coordinates



0 BEGIN PGM CIRCULAR MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Define blank form for graphic workpiece simulation
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 CYCL DEF 1.0 GENERATOR	Cycle GENERATOR (see "Cycle 1 GENERATOR" on page 133)
4 CYCL DEF 1.1 P-TAB CUST1	Select erosion table (here, table CUST1)
5 CYCL DEF 1.2 MAX=6 MIN=6	Set power stage (here, to stage 6)
6 TOOL DEF 6 L+0 R+10	Define electrode in the program
7 TOOL CALL 6 Z U+1.5	Call electrode in the infeed axis Z, undersize 1.5 mm
8 L Z+100 C+0 R0 F MAX M37	Retract in the infeed axis; orient electrode; eroding OFF
9 L X-10 Y-10 R F MAX	Pre-position in X and Y; rapid traverse
10 L Z-5 R0 F MAX M	Move to working depth
11 L X+5 Y+5 RL F M36	Approach the contour at point 1 with radius compensation; eroding ON
12 L X+5 Y+85 R F M	Point 2: first straight line for corner 2
13 RND R10 F	Insert radius with R = 10 mm
14 L X+30 Y+85 R F M	Move to point 3: Starting point of the arc with CR
15 CR X+70 Y+95 R+30 DR-	Move to point 4: End point of the arc with CR, radius 30 mm
16 L X+95 R F M	Move to point 5
17 L X+95 Y+40 R F M	Move to point 6
18 CT X+40 Y+5 R F M	Move to point 7: End point of the arc, radius with tangential
	connection to point 6, TNC automatically calculates the radius

`19 L X+5 R F M`	Move to last contour point 1
`20 L X-10 Y-10 R F M37`	Retract tool in the working plane; eroding OFF
`21 L Z+100 R0 F MAX`	Move electrode to set-up clearance; rapid traverse
`22 END PGM CIRCULAR MM`	

6.5 Path Contours — Polar Coordinates

Overview

With polar coordinates you can define a position in terms of its angle PA and its distance PR relative to a previously defined pole CC (see "Polar coordinates" on page 45).

Polar coordinates are useful with:

- Positions on circular arcs
- Workpiece drawing dimensions in degrees, e.g. bolt hole circles

Polar coordinates are identified with a P.

Overview of path functions with polar coordinates

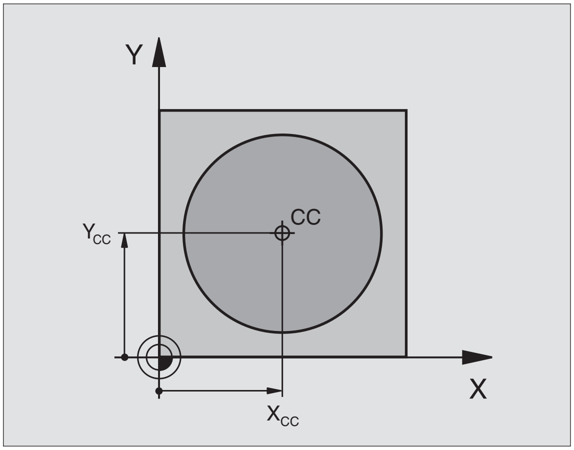
Function	Path function key	Tool movement	Required input
Line **LP**	L + P	Straight line	Polar radius, polar angle of the straight-line end point
Circular arc **CP**	C + P	Circular path around circle center/ pole CC to arc end point	Polar angle of the arc end point, direction of rotation
Circular arc **CTP**	CT + P	Circular arc with tangential connection to the preceding contour element	Polar radius, polar angle of the arc end point
Helical interpolation	C + P	Combination of a circular and a linear movement	Polar radius, polar angle of the arc end point, coordinate of the end point in the tool axis

Polar coordinate origin: Pole CC

You can define the pole CC anywhere in the part program before blocks containing polar coordinates. Enter the pole in Cartesian coordinates as a circle center in a CC block.

CC

▶ `Coordinates` CC: Enter Cartesian coordinates for the pole, or
If you want to use the last programmed position, do not enter any coordinates. Before programming polar coordinates, define the pole CC. You can only define the pole CC in Cartesian coordinates. The pole CC remains in effect until you define a new pole CC.

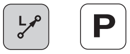


Example NC blocks

```
12 CC X+45 Y+25
```

Straight line LP

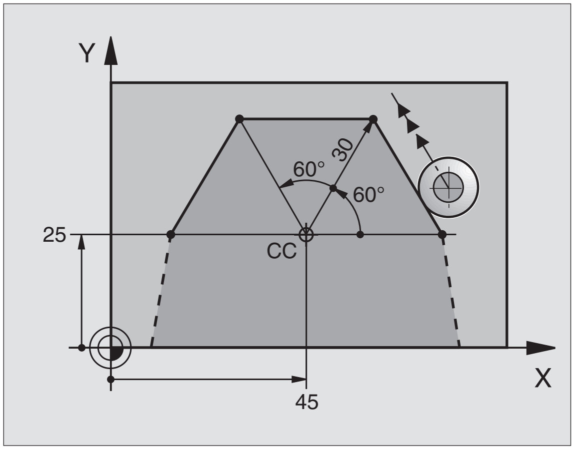
The electrode moves in a straight line from its current position to the straight-line end point. The starting point is the end point of the preceding block.



- **Polar coordinates radius PR**: Enter the distance from the pole CC to the straight-line end point.
- **Polar coordinates angle PA**: Angular position of the straight-line end point between –360° and +360°.

The sign of PA depends on the angle reference axis:

- Angle from angle reference axis to PR is counterclockwise: PA>0
- Angle from angle reference axis to PR is clockwise: PA<0

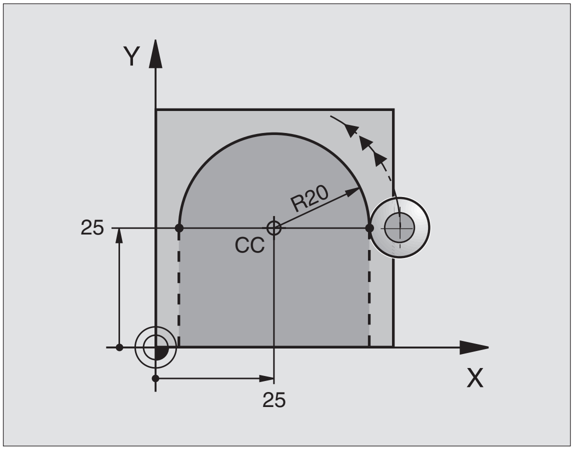


Example NC blocks

```
12 CC X+45 Y+25
13 LP PR+30 PA+0 RR F M
14 LP PR PA+60 R F M
15 LP PR IPA+60 R F M
16 LP PR PA+180 R F M
```

Circular path CP around pole CC

The polar coordinate radius PR is also the radius of the arc. It is defined by the distance from the starting point to the pole CC. The last programmed electrode position before the CP block is the starting point of the arc.



> If you are using an electrode with tool compensation in the XY plane, you must rotate the electrode in synchrony with the angle on circular arcs. For example, for a semicircle you must rotate the C axis by 180° (incremental).

- **`Polar coordinates angle PA:`** Angular position of the arc end point
- **`Direction of rotation DR`**

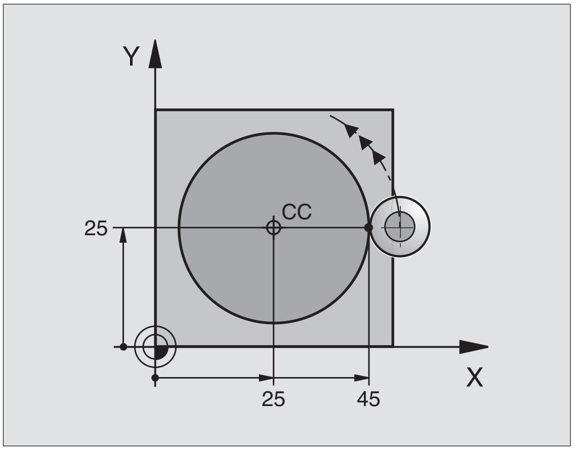
Example NC blocks

```
18 CC X+25 Y+25
19 LP PR+20 PA+0 RR F M
20 CP PA+180 DR+ R F M
```

> - For incremental coordinates, enter the same sign for DR and PA.
> - For PA you may enter values from –5400 to +5400.
> - The end point of the circle may **not** be identical with the starting point of the circle.

Full circle

For a full circle you must program the incremental polar coordinate angle IPA with 360°. The electrode moves from the starting point around the circle center CC.
The linear coordinate IC +360 rotates the electrode in synchrony with the angle on the circular path.



> You can only program a full circle with the incremental polar coordinate angle IPA.

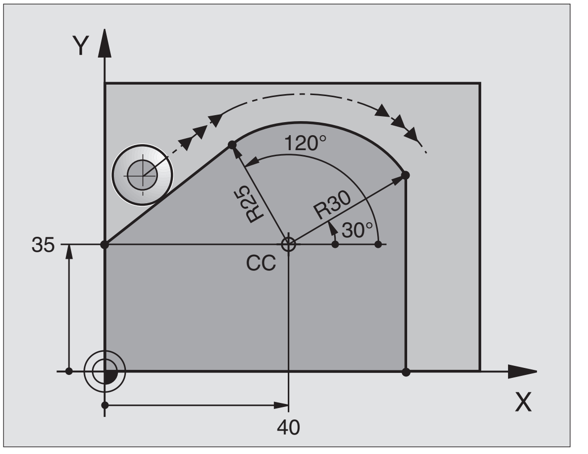
Example NC blocks

```
5 CC X+25 Y+25
6 L X+45 Y+25 RR F M
7 CP IPA +360 IC+360 DR+ R F M
```

Circular path CTP with tangential connection

The tool moves on a circular path, starting tangentially from a preceding contour element.

- **`Polar coordinates radius PR:`** Distance from the arc end point to the pole CC
- **`Polar coordinates angle PA:`** Angular position of the arc end point



Example NC blocks

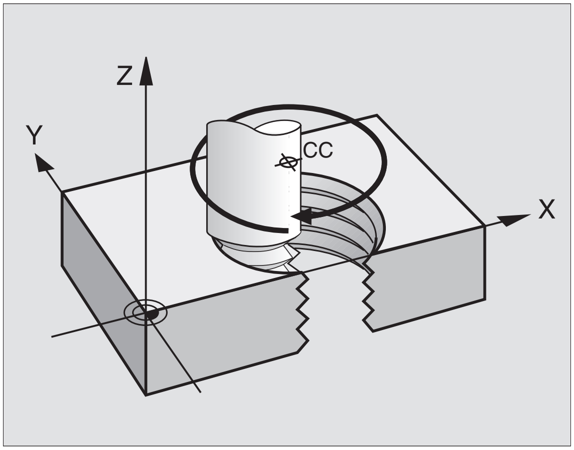
```
12 CC X+40 Y+35
13 L X+0 Y+35 RL F M
14 LP PR+25 PA+120 R F M
15 CTP PR+30 PA+30 R F M
16 L Y+0 R F M
```

The pole CC is **not** the center of the contour arc!

Helical interpolation

A helix is a combination of a circular movement in a main plane and a liner movement perpendicular to this plane.

A helix is programmed only in polar coordinates.



Application

- Large-diameter internal and external threads
- Lubrication grooves

Calculating the helix

To program a helix, you must enter the total angle through which the tool is to move on the helix in incremental dimensions, and the total height of the helix.

For calculating a helix that is to be cut in a upward direction, you need the following data:

Thread revolutions n	Thread revolutions + thread overrun at the start and end of the thread
Total height h	Thread pitch P times thread revolutions n
Incremental total angle IPA	Number of revolutions times 360° + angle for beginning of thread + angle for thread overrun
Starting coordinate Z	Pitch P times (thread revolutions + thread overrun at start of thread)

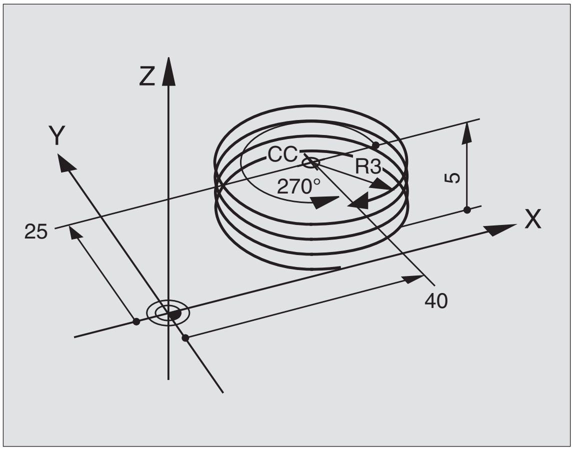
Shape of the helix

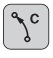
The table below illustrates in which way the shape of the helix is determined by the work direction, direction of rotation and radius compensation.

Internal thread	Work direction	Direction	Radius comp.
Right-handed Left-handed	Z+ Z+	DR+ DR–	RL RR
Right-handed Left-handed	Z– Z–	DR DR+	RR RL
External thread			
Right-handed Left-handed	Z+ Z+	DR+ DR–	RR RL
Right-handed Left-handed	Z– Z–	DR DR+	RL RR

Programming a helix

- Always enter the same algebraic sign for the direction of rotation DR and the incremental total angle IPA. The tool may otherwise move in a wrong path and damage the contour.
- For the total angle IPA, you can enter values from –5400° to +5400°. If the thread has more than 15 revolutions, program the helix in a program section repeat
(see "Program Section Repeats" on page 176)
- If you are using an electrode with tool compensation in the XY plane, you must rotate the electrode in synchrony with the angle on circular arcs. Enter the same angle in incremental dimensions for the C axis as for the total angle.



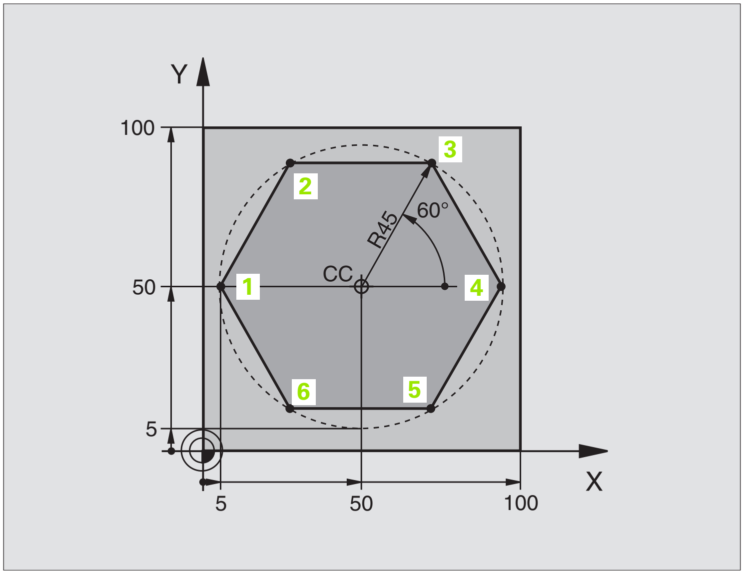


- **Polar coordinates angle:** Enter the total angle of tool traverse along the helix in incremental dimensions. **After entering the angle, identify the tool axis with an axis selection key.**
- **Coordinate**: Enter the coordinate for the height of the helix in incremental dimensions.
- Enter the coordinate for the angle-synchronous rotation of the electrode in incremental dimensions, e.g., IC -1800.
- **Direction of rotation DR**
Clockwise helix: DR–
Counterclockwise helix: DR+
- **Radius compensation RL/RR/R0**
Enter the radius compensation according to the table above.

Example NC blocks: Thread M6 x 1 mm with 5 revolutions

```
12 CC X+40 Y+25
13 L Z+0 R F M37
14 LP PR+3 PA+270 RL F M
15 CP IPA-1800 IZ+5 IC-1800 DR- R F M
```

Example: Linear movement with polar coordinates



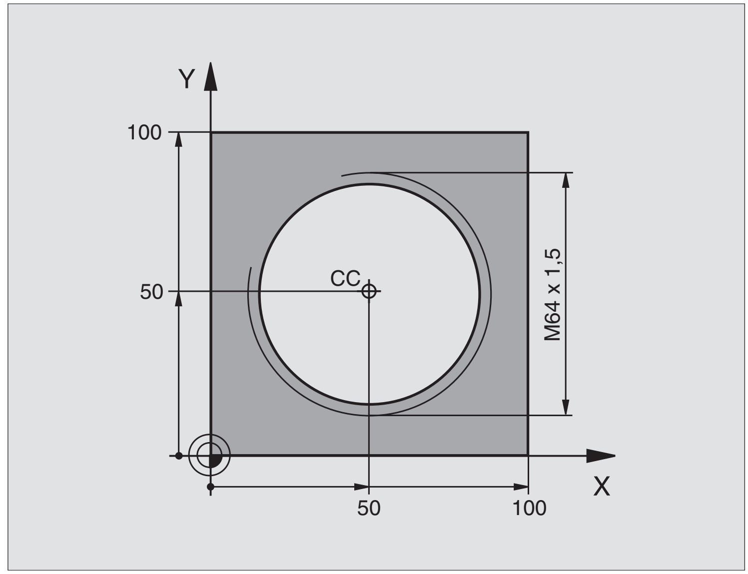
0 BEGIN PGM LINEARPO MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Define the workpiece blank
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 CYCL DEF 1.0 GENERATOR	Cycle GENERATOR (see "Cycle 1 GENERATOR" on page 133)
4 CYCL DEF 1.1 P-TAB CUST1	Select erosion table (here, table CUST1)
5 CYCL DEF 1.2 MAX=6 MIN=6	Set power stage
6 TOOL DEF 6 L+0 R+15	Define electrode in the program
7 TOOL CALL 6 Z U+1.5	Call electrode in the infeed axis, undersize 1.5 mm
8 CC X+50 Y+50	Define the datum for polar coordinates
9 L Z+100 C+0 R0 F MAX M37	Retract in the infeed axis; orient electrode; eroding OFF
10 LP PR+80 PA-190 R0 F MAX	Pre-position in X and Y; rapid traverse
11 L Z-10 R0 F M	Move to working depth
12 LP PR+45 PA+180 RL M36	Approach the contour at point 1 with radius compensation; eroding ON
13 LP PR PA+120 R F M	Move to point 2
14 LP PR PA+60 R F M	Move to point 3
15 LP PR PA+0 R F M	Move to point 4
16 LP PR PA-60 R F M	Move to point 5
17 LP PR PA-120 R F M	Move to point 6
18 LP PR PA+180 R F M	Move to point 1
19 LP PR+80 PA+170 R0 F MAX M37	Retract tool in the working plane; eroding OFF

`20  L Z+100 R0 F MAX M`	Move electrode to set-up clearance; rapid traverse
`21 END PGM LINEARPO MM`	

Example: Helix

Right-handed internal thread M64 x 1.5 with starting angle 0°, end angle 360° and 8 revolutions n_G. The thread overrun is 0.5 at both the start of thread n_S and end of thread n_E.

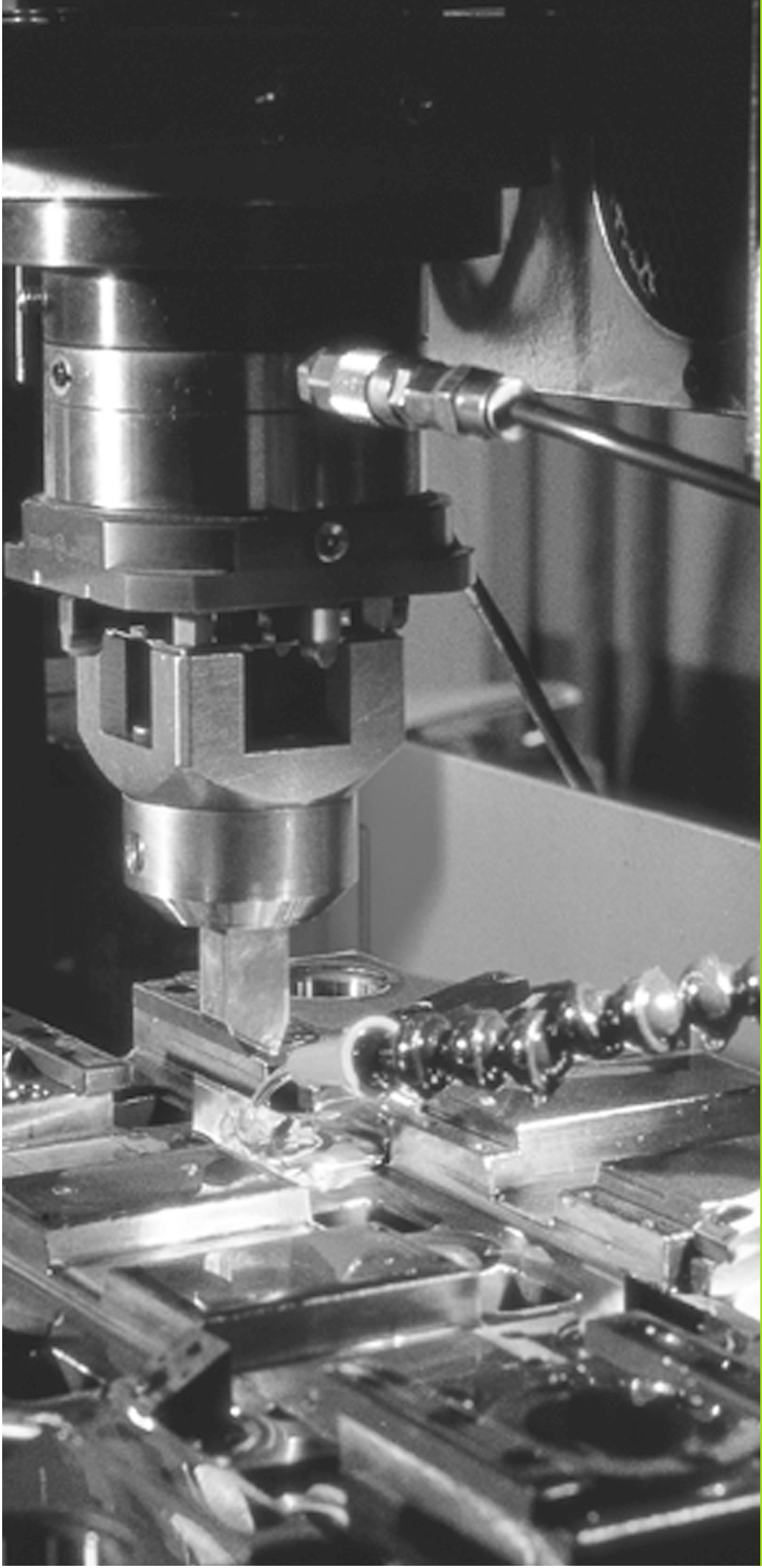
The calculation of the entered values is explained in ”Calculating the helix” on page 113.



`0 BEGIN PGM HELIX MM`	
`1 BLK FORM 0.1 Z X+0 Y+0 Z-20`	Define the workpiece blank
`2 BLK FORM 0.2 X+100 Y+100 Z+0`	
`3 CYCL DEF 1.0 GENERATOR`	Cycle GENERATOR (see ”Cycle 1 GENERATOR” on page 133)
`4 CYCL DEF 1.1 P-TAB HDH700`	Select erosion table (here, table HDH700)
`5 CYCL DEF 1.2 MAX=6 MIN=6`	Set power stage
`6 TOOL DEF 6 L+0 R+5`	Define electrode in the program
`7 TOOL CALL 6 Z U+1.5`	Call electrode in the infeed axis Z, undersize 1.5 mm
`8 L Z+100 C0 R0 F MAX M`	Retract in the infeed axis; orient electrode
`9 L X+50 Y+50 R0 F MAX M`	Pre-position in X and Y; rapid traverse
`10 CC`	Transfer the last programmed position as the pole
`11 L Z-12.75 R F MAX M`	Move to working depth
`12 LP PR+32 PA–180 RL F M36`	Move to first contour point with radius compensation; eroding ON
`13 CP IPA+3240 IZ+13.5 IC+3240 DR+ R F M`	Helical interpolation; also rotate electrode in angle synchronicity
`14 L X+50 Y+50 R0 F MAX M37`	Retract tool in the working plane; eroding OFF
`15 L Z+100 F MAX`	Move electrode to set-up clearance; rapid traverse
`16 END PGM HELIX MM`	

To cut a thread with more than 16 revolutions

`...`	
`11 L Z-12.75 R0 F M`	
`12 LP PR+32 PA-180 RL M36`	
`13 LBL 1`	Identify beginning of program section repeat
`14 CP IPA+360 IZ+1.5 IC+360 DR+ R F M`	Enter the thread pitch as an incremental IZ dimension
`15 CALL LBL 1 REP 24`	Program the number of repeats (thread revolutions)
`16 L X+50 Y+50 R0 F MAX M37`	



7

Programming: Miscellaneous functions

7.1 Entering Miscellaneous Functions M and STOP

Fundamentals

With the TNC’s miscellaneous functions – also called M functions – you can affect:

- Program run
- Machine functions
- Electrode behavior

An overview of how the miscellaneous functions are set in the TNC is shown on the inside rear cover. This table shows if a function becomes effective at the beginning or at the end of the block in which it was programmed.

Answer the dialog question in the positioning block:

M — To enter the miscellaneous function, press the M soft key.

MISCELLANEOUS FUNCTION M ?

38 ENT — Enter miscellaneous function, e.g. M38.

Entering an M function in a STOP block

MISCELLANEOUS FUNCTION M ?

39 ENT — Enter miscellaneous function, e.g. M39.

Example NC block

```
7  STOP M39
```

If you program a miscellaneous function in a STOP block, the program run is interrupted at the block.

Certain miscellaneous functions do not work on certain machines. There may also be additional miscellaneous functions available which have been defined by the machine tool builder.

The program run or test run is interrupted when the NC block containing the STOP function is reached.
You can also enter an M function in a STOP block.
If the program run is to be interrupted for a specified time, use Cycle 9 DWELL TIME (see also "DWELL TIME (Cycle 9)" on page 171).

Enter STOP function

STOP — Select STOP function.

MISCELLANEOUS FUNCTION M ?

6 ENT — If desired: Enter miscellaneous function, e.g. M6 (electrode change).

Example NC block

```
7 STOP M6
```

7.2 Miscellaneous Functions for Program Run Control, Electrode and Flushing

Overview

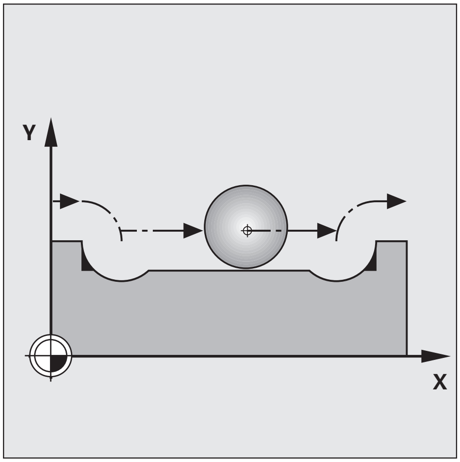
M	Effect	Effective at block start	end
M00	Stop program run		■
M02	Stop program run Go to block 1 Clear the status display (dependent on machine parameter 7300)		■
M03	Free rotation of the C axis (direction of rotation set by the machine tool builder)	■	
M04	Free rotation of the C axis (direction of rotation set by the machine tool builder)	■	
M05	Stop free rotation of the C axis		■
M06	Electrode changing Program run stop (dependent on machine parameter 7440)		■
M08	Flushing ON	■	
M09	Flushing OFF		■
M13	Functionality of M03 + M08	■	
M14	Functionality of M04 + M08	■	
M30	Same as M02		■

7.3 Miscellaneous Functions for Contouring Behavior and Coordinate Data

Introduction

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

- Machining small contour steps
- Machining open contours
- Entering machine-referenced coordinates
- Retracting the electrode to the block starting point at the end of block



Machining small contour steps: M97

Standard behavior (without M97)

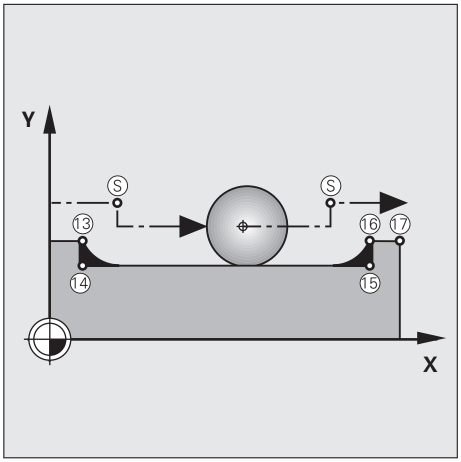
The TNC inserts a transition arc at outside corners. If the contour steps are very small, however, the tool would damage the contour.

In such cases the TNC interrupts program run and generates the error message "Tool radius too large".

Behavior with M97

The TNC calculates the intersection of the contour elements—as at inside corners—and moves the tool over this point (see illustration bottom right).

Program M97 in the same block as the outside corner.



Effect

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

> A corner machined with M97 will not be completely finished. You may wish to rework the contour with a smaller tool.

Example NC blocks

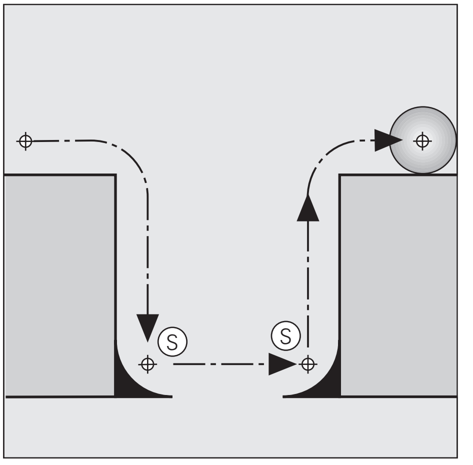
`5 TOOL DEF L ... R+20`	Large tool radius
`...`	
`13 L X ... Y ... R.. F .. M97`	Move to contour point 13
`14 L IY-0.5 .... R .. F..`	Machine small contour step 13 to 14
`15 L IX+100 ...`	Move to contour point 15
`16 L IY+0.5 ... R .. F.. M97`	Machine small contour step 15 to 16
`17 L X .. Y ...`	Move to contour point 17

Machining open contours: M98

Standard behavior (without M98)

The TNC calculates the intersections of the electrode paths at inside corners 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.



Behavior with M98

With the miscellaneous function M98, the TNC temporarily suspends radius compensation to ensure that both corners are completely machined.

Effect

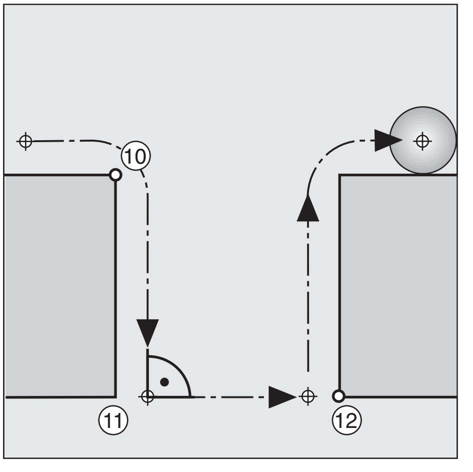
M98 is effective only in the blocks in which it is programmed.

M98 takes effect at the end of block.

Example NC blocks

Move to the contour points 10, 11 and 12 in succession:

```
10 L X ... Y... RL F
11 L X... IY-... M98
12  L IX+ ...
```



Programming machine-referenced coordinates: M91/M92

Scale reference point

The scales are provided with one or more reference marks. A reference mark indicates 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, the scale reference point is the position of the left-most reference mark (at the beginning of the measuring range).

Machine datum

The machine datum is required for the following tasks:

- Defining the limits of traverse (software limit switches)
- Moving to machine-referenced positions (such as tool change positions)
- Setting the workpiece datum

The distance in each axis from the scale reference point to the machine datum is defined by the machine tool builder in a machine parameter.

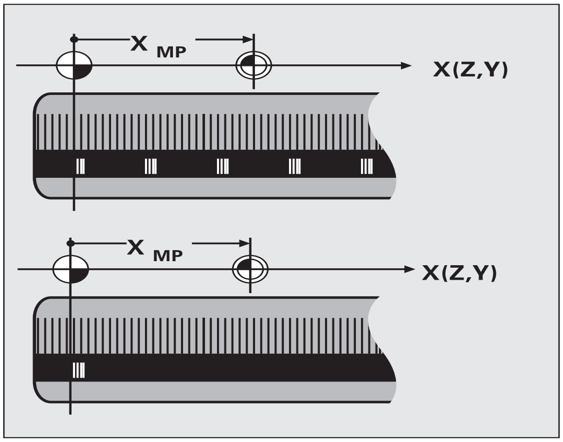
Standard behavior

The TNC references coordinates to the workpiece datum.

Behavior with M91 – Machine datum

If you want the coordinates in a positioning block to be referenced to the machine datum, end the block with M91.

The coordinate values on the TNC screen are referenced to the machine datum. Switch the display of coordinates in the status display to REF (see also "Status Display" on page 9).



Behavior with M92 – Additional machine datum

In addition to the machine datum, the machine tool builder can also define an additional machine-based position as a reference point.

For each axis, the machine tool builder defines the distance between the machine datum and this additional machine datum. Refer to the machine manual for more information.

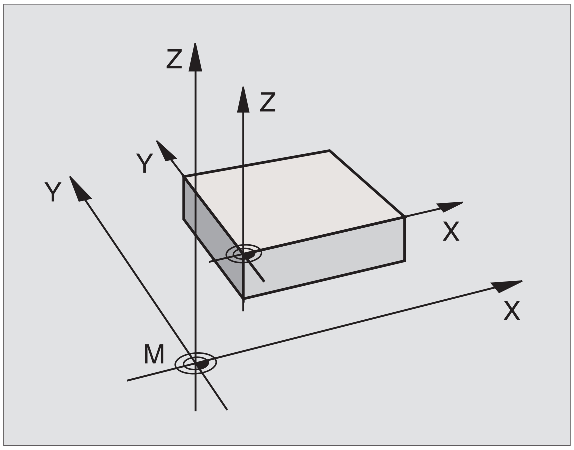
If you want the coordinates in a positioning block to be based on the additional machine datum, end the block with M92.

Radius compensation remains the same in blocks that are programmed with M91 or M92. The tool length, however, is **not** compensated.

Effect

M91 and M92 are effective only in the blocks in which they are programmed.

M91 and M92 take effect at the start of block.



Workpiece datum

The position of the datum for the workpiece coordinates is defined in the MANUAL OPERATION mode (see also "Datum Setting" on page 22). The user enters the coordinates of the datum for workpiece machining in this mode.

Retracting electrode to block starting point at end of block: M93

Standard behavior

The TNC executes the NC blocks as programmed.

Behavior with M93

The TNC retracts the electrode at the end of a block and moves it back to the starting point of this block. This function can be used not only with linear but also with circular and helical movements.

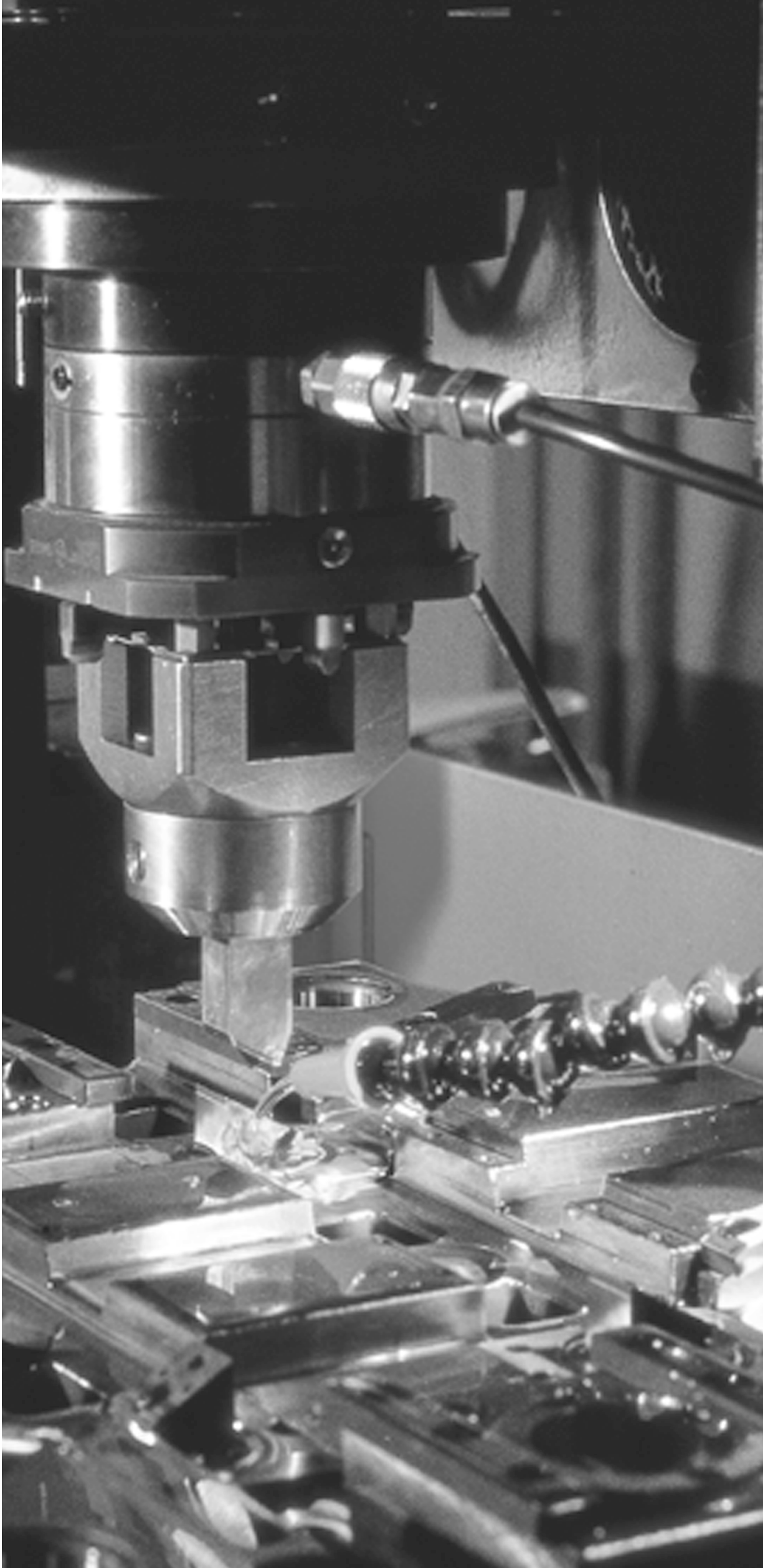
M93 is effective only in the blocks in which it is programmed, and only if M36 (eroding ON) is active.

7.4 Vacant miscellaneous functions

Vacant miscellaneous functions are defined by the machine tool builder. They are described in your machine manual.

M	Function	Effective at block start	Effective at block end	M	Function	Effective at block start	Effective at block end
M01			■	M52			■
M07		■		M53			■
M10			■	M54			■
M11		■		M55		■	
M12			■	M56		■	
M15		■		M57		■	
M16		■		M58		■	
M17		■		M59		■	
M18		■		M60			■
M19			■	M61		■	
M20		■		M62		■	
M21		■		M63			■
M22		■		M64			■
M23		■		M65			■
M24		■		M66			■
M25		■		M67			■
M26		■		M68			■
M27		■		M69			■
M28		■		M70			■
M29		■		M71		■	
M31		■		M72		■	
M32			■	M73		■	
M33			■	M74		■	
M34			■	M75		■	
M35			■	M76		■	
M40		■		M77		■	

M	Function	Effective at block		M	Function	Effective at block	
		start	end			start	end
M41		■		M78		■	
M42		■		M79		■	
M43		■		M80		■	
M44		■		M81		■	
M45		■		M82		■	
M46		■		M83		■	
M47		■		M84		■	
M48		■		M85		■	
M49		■		M86		■	
M50		■		M87		■	
M51		■		M88		■	



8

Programming: Cycles

8.1 General Overview of Cycles

Frequently recurring machining sequences which comprise several working steps are stored in the TNC as standard cycles. To enter the required data the user only has to respond to questions asked by the TNC.

Cycles are divided into the following groups:

- **GENERATOR**, for basic information on the eroding process.
- **CONTOUR**, for machining closed contours.
- **DISK**, which allows you to easily carry out many different tasks, and **EROSION TIME LIMIT**, which depends on the DISK cycle.
- **TOOL DEF**, which allows you to define electrodes with compensation values.
- **Coordinate transformation cycles** for shifting, rotating, mirroring, enlarging and reducing contours.
- **Special cycles:** dwell time and program call.

Prerequisites

Before a cycle call you must have programmed:

- BLK FORM for graphic display
- Electrode call
- Positioning block for starting position X, Y
- Positioning block for starting position Z (set-up clearance)

Start of effect

All cycles except PGM CALL go into effect as soon as they are defined. PGM CALL must be called.

Dimensions in the electrode axis

Infeeds in the electrode axis always refer to the position of the electrode at the moment the cycle is called. The TNC interprets the coordinates as incremental dimensions; you do not have to press the I key.

OEM cycles

The machine tool builder can prepare additional cycles and store them in the TNC's ROM. These cycles can be called with the cycle numbers 30 to 99. Refer to your machine tool manual.
The control goes to the first available OEM cycle when the GOTO OEM CYCLE soft key is pressed.

Programming a cycle

Press the CYCL DEF key to open the cycle directory. Select the desired cycle and program it in the dialog. Using the DISK cycle as an example, the flow chart illustrates how any cycle can be defined:

CYCL DEF — Open the cycle directory.

CYCL DEF 1 GENERATOR

↓ ↑ — Select, for example, Cycle 17 with the vertical arrow keys.

GOTO OEM CYCLE — The control goes to the first available OEM cycle when the GOTO OEM CYCLE soft key is pressed.

GOTO □ 17 — Address the desired cycle directly with GOTO.

ENT — Confirm your entry with the ENT key.

CYCL DEF 17 DISK

ENT — Open selected cycle

ERODING AXIS AND DEPTH ?

Z -5 — Enter the eroding axis and depth, e.g. Z = –5 mm.

ENT — Confirm your entry with the ENT key.

MISCELLANEOUS FUNCTION M ?

36 ENT — Enter a miscellaneous function, e.g. M36 (eroding ON).

EXPANSION RADIUS ?

0 ENT — Enter expansion mode, e.g. 0.

Example NC blocks

```
17.0  DISK
17.1  Z-5, M36
17.2  RAD=75, MOD=0
```

8.2 Cycle 1 GENERATOR

Working with an erosion table

If you want to work with erosion tables in a program, you must copy Cycle 1.0 GENERATOR into the program.

Program the following information in this cycle:

- Which erosion table P-TAB you want to use
- The maximum power stage MAX for subsequent machining
- The minimum power stage MIN for subsequent machining

In a program run operating mode the TNC displays the highest and lowest power stage after the GENERATOR cycle has been executed.

Working without an erosion table

If you are not working with an erosion table, do not copy Cycle 1.0 GENERATOR into the program. In this case you must enter the erosion parameters in Q parameters Q90 to Q99.

To enter Cycle 1.0 GENERATOR

CYCL DEF — Open the cycle directory.

CYCL DEF 1 GENERATOR

ENT — Open selected cycle.

EROSION TABLE ?

5 ENT — Enter name of the erosion table, e.g. 5.

MAX. POWER STAGE ?

15 ENT — Enter the highest power stage for machining, e.g. 15.

MIN. POWER STAGE ?

2 ENT — Enter the lowest power stage for machining, e.g. 2.

Example NC blocks

```
1.0  GENERATOR
1.1  P-TAB 5
1.2  MAX=15, MIN=2
```

Changing the power stage

The TNC stores the current power stage in Q parameter Q99. If you want to change the power the stage, assign to Q99 the value of the new power stage.

Example NC block

Desired power stage = 12

```
FN 0: Q99 = 12
```

8.3 Electrode Definition

Cycle 3 TOOL DEF

Just as in the NC block TOOL DEF, you can define the number and radius of an electrode in Cycle 3 TOOL DEF. In addition you can enter a tool compensation value.

In Cycle 3 TOOL DEF you enter the

- Tool number T from 1 to 9 999
- Tool radius R in mm (R > 0)
- Tool compensation for up to four axes (in mm)

Sign for tool compensation

- To compensate the tool from the tool datum in the direction of the positive coordinate axis: compensation value > 0
- To compensate the tool from the tool datum in the direction of the negative coordinate axis: compensation value < 0

Determine the compensation values for Cycle 3 TOOL DEF at 0° angular position so that compensation will be activated with the correct values in the working plane when the C axis is rotated.

To enter Cycle 3 TOOL DEF

CYCL DEF — Open the cycle directory.

CYCL DEF 1 GENERATOR

GOTO 3 — Go to Cycle 3.0 TOOL DEF.

ENT — Confirm your entry with the ENT key.

CYCL DEF 3 TOOL DEF

ENT — Select Cycle 3.0 TOOL DEF.

TOOL NUMBER ?

5 ENT — Enter tool number, e.g. T = 5.

TOOL RADIUS ?

10 ENT — Enter tool radius, e.g. R = 10 mm.

TOOL COMP. ?

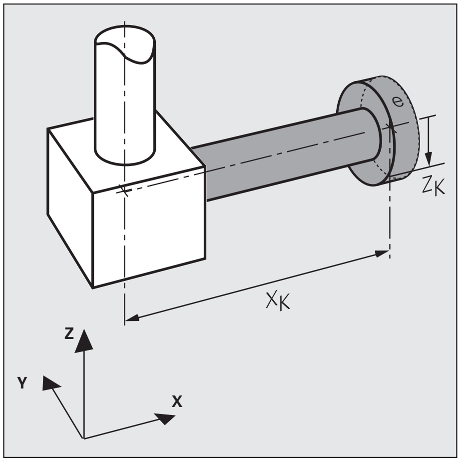
X -10 — Enter the coordinate axes and compensation values (for example, X = – 10 mm).

ENT — Confirm your entry with the ENT key.

Z -5 — Enter the coordinate axes and compensation values (for example, Z = – 5 mm).

ENT — Confirm your entry with the ENT key.

END — Press END when you have entered all compensation values.



Example NC blocks

```
3.0  CYCL DEF   TOOL DEF
3.1  CYCL DEF   T=5   R+10
3.2  CYCL DEF   X-10  Z-5
```

Circular path with electrode compensation

If you enter an electrode compensation, you must rotate the electrode **in synchrony with the angle** on circular arcs. For example, for a semicircle you must rotate the C axis by 180°.

8.4 Erosion Cycles

Overview

The TNC offers five erosion cycles:

- Cycle 14 CONTOUR GEOMETRY
- Cycle 16 ORBIT
- Cycle 17 DISK
- Cycle 2 ERO.TIME LIM.
- Cycle 4 SPARK-OUT TIME

Cycle 14 CONTOUR GEOMETRY

The CONTOUR GEOMETRY cycle is a machining cycle. You use it to cyclically erode a closed contour in the working plane with the programmed feed rate. Gap control is effective in the eroding axis which you defined in the cycle. You define the contour to be eroded in a separate program. After the programmed eroding depth is reached and the defined sparking-out distance is traversed, the TNC ends the erosion cycle. The electrode does not retract automatically.

In Cycle 14 CONTOUR GEOMETRY you enter the

- Eroding axis
- Eroding depth
- Miscellaneous function M
- Contour program PGM
- Sparking-out distance in percent PRC

If necessary, you may also use Q parameters for the cycle definition of the eroding depth and the sparking-out distance.

Eroding axis and depth

The eroding axis determines the coordinate axis parallel to which eroding takes place in the ”depth.”

The **sign** of the eroding depth determines whether the working direction is the direction of the positive coordinate axis (depth +) or of the negative coordinate axis (depth –).

You can enter the eroding depth in **absolute** or **incremental** dimensions.

Miscellaneous function M

You can enter a miscellaneous function in Cycle 14 CONTOUR GEOMETRY, such as M36 (eroding ON).

Contouring program PGM

The cycle parameter PGM determines the contouring program that is to be used by the TNC.

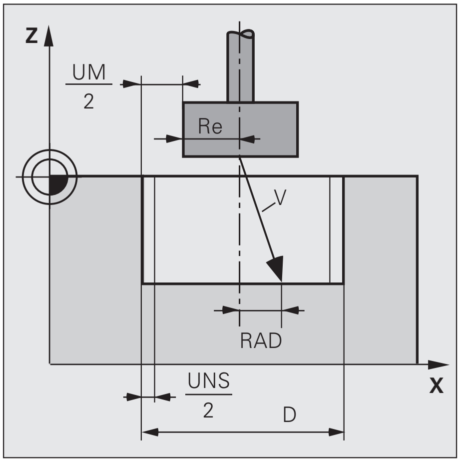
Requirements for the contouring program:

- The contour programmed in the contouring program must be a closed contour, such as a rectangle.
- The start of the contour should be in the middle, so that the contour is scalable with Cycle 11.
- No coordinates may be programmed for the eroding axis in the contouring program. The eroding axis and eroding depth are specified in Cycle 14.
- No M functions that cause a geometry reset, such as M02 or M30, may be programmed in the contouring program.

Sparking-out distance in percent PRC

This parameter determines how far the TNC should retract for sparking-out after having reached the eroding depth. The value is to be entered as a percentage of the total contour length.

Cycle 16 ORBIT



The ORBIT cycle is a machining cycle which facilitates programming of spark-out behavior and movement of the electrode.

In Cycle 16 ORBIT you enter the

- Eroding axis
- Eroding depth
- Miscellaneous function M
- Expansion radius RAD
- Rotational direction DIR
- Expansion mode PAT
- Spark-out mode SPO

If necessary, you may also use Q parameters for the cycle definition.

Eroding axis and depth

The eroding axis determines the coordinate axis parallel to which eroding takes place in the "depth."

The **sign** of the eroding depth determines whether the working direction is the direction of the positive coordinate axis (depth +) or of the negative coordinate axis (depth –).

You can enter the eroding depth in **absolute** or **incremental** dimensions.

Miscellaneous function M

You can enter a miscellaneous function in Cycle 16 ORBIT, such as M36 (eroding ON).

Expansion radius RAD

The TNC feeds the electrode in radial direction (perpendicular to the eroding depth) by the value of the expansion radius.

> The electrode radius Re must be larger than the expansion radius RAD. Otherwise the pocket (disk) will not be completely eroded.

Calculating the expansion radius RAD

If the diameter D of the disk is known, you can calculate the expansion radius RAD from the following data:

- Diameter D of the disk
- Electrode undersize UM
- Electrode minimum undersize UNS
- Electrode radius Re

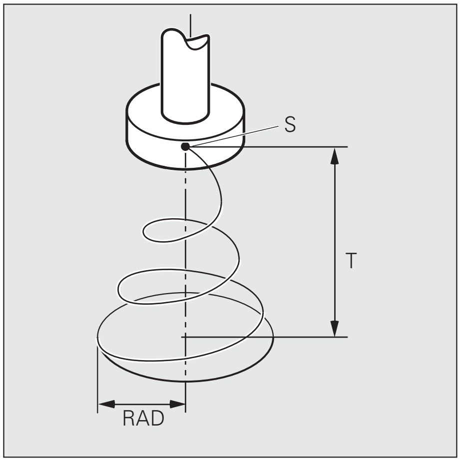
RAD = 0.5 • (UM – UNS) **= 0.5 • D – Re – 0.5 • UNS**

Rotational direction DIR

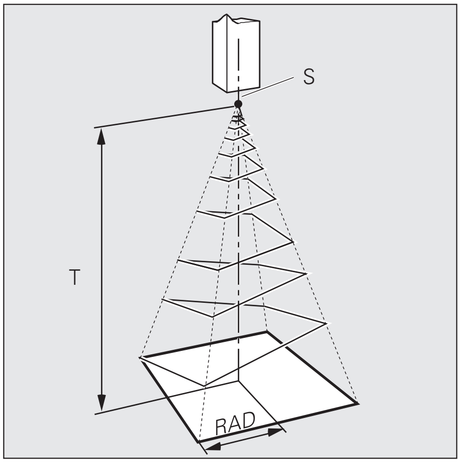
Counterclockwise erosion movement: DIR = 0
Clockwise erosion movement: DIR = 1

Expansion mode PAT

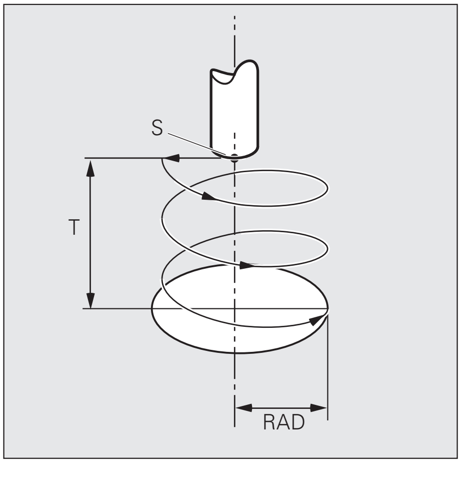
The expansion mode PAT determines the movement of the electrode during erosion.



- **PAT = 0: Circular expansion (top illustration)**
  From the starting depth S the electrode moves along the surface of a circular cone until it reaches the programmed eroding depth T and the expansion radius RAD. The gap is controlled along an angular vector. The electrode is retracted to the starting point along a diagonal path.
- **PAT = 1: Quadratic expansion (center illustration)**
  Same as PAT = 0, but with quadratic expansion instead of circular expansion.
- **PAT = 2: Circular orbital sinking (bottom illustration)**
  The electrode moves from the starting point S by the expansion radius RAD in radial direction. It then follows a circular path until reaching the eroding depth. The gap is controlled only in the eroding axis. The electrode is retracted to the starting point along a diagonal path.
- **PAT = 3: Quadratic orbital sinking**
  Same as PAT = 2, but with quadratic sinking instead of circular sinking.



- **PAT = 4: Circular expansion in two phases**
  1.) From the starting depth S the electrode moves along the surface of a circular cone (0° direction) until it reaches the programmed eroding depth T and the expansion radius RAD. The gap is controlled along an angular vector.
  2.) At the eroding depth T, expansion is carried out in a circular path with radius = entered end radius. The gap is controlled along the circular path. The electrode is retracted first along the erosion path and then diagonally back to the starting point.
- **PAT = 5: Quadratic expansion in two phases**
  Same as PAT = 4, but with quadratic expansion instead of circular expansion.
- **PAT = 6: Circular expansion in two phases**
  1.) From the starting depth S the electrode moves along the surface of a circular cone (0° direction) until it reaches the programmed eroding depth T and the expansion radius RAD. The gap is controlled along an angular vector.
  2.) At the eroding depth T, expansion is carried out in a circular path with radius = entered end radius. The gap is controlled along the circular path. The electrode is retracted to the starting point along a diagonal path.
- **PAT = 7: Quadratic expansion in two phases**
  Same as PAT = 6, but with quadratic expansion instead of circular expansion.



There is the danger of collision if retraction to the starting point follows a diagonal vector.
Select an electrode radius Re greater than the expansion radius RAD for the corresponding expansion modes.

Spark-out mode SPO

The spark-out mode SPO determines the manner and duration of the spark-out.

- **SPO = 0: Fast sparking-out**
  Spark-out depends on the end radius and machine parameter MP2110, or, if Cycle 4 SPARK-OUT is defined, on the parameters in Cycle 4.
- **SPO = 1: Sparking-out**
  Spark-out begins when the end radius has been reached, and the electrode has been in free run for 1.25 orbits.

Feed rates for eroding with Cycle 14 ORBIT

The **feed rate for rotary motion** is the same as the last-programmed feed rate. It is limited by user parameters MP1092 to MP1097.

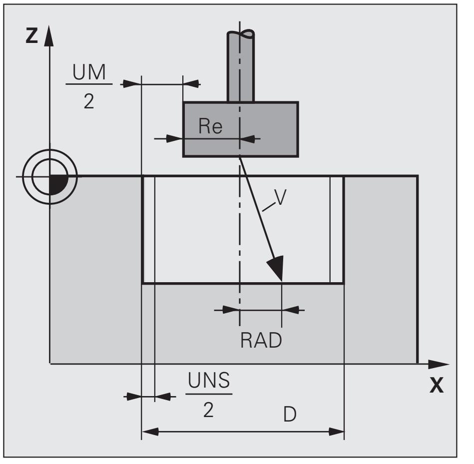
The **feed rate in the tool axis direction** is determined by the gap control.

Standard behavior with short circuit

In the event of a short circuit, the electrode is stopped and retracted along the infeed vector.
Once the short circuit is eliminated, the TNC moves the electrode back along the infeed vector toward the workpiece but stops a certain distance before the point where the short circuit occurred (this distance is defined in parameter MP2050).

The machine tool builder may have specified a different retraction behavior in the event of short circuiting than is described here.
Refer to your machine tool manual.

Cycle 17 DISK



The DISK cycle is a machining cycle. It facilitates the sparking-out behavior and movement of the electrode.
You can use the DISK cycle to develop machining sequences such as for conical cavities (see Chapter 7).

In Cycle 17 DISK you enter the

- Eroding axis
- Eroding depth
- Miscellaneous function M
- Expansion radius RAD
- Expansion mode MOD

If necessary, you may also use Q parameters for the cycle definition.

Eroding axis and depth

The eroding axis determines the coordinate axis parallel to which eroding takes place in the "depth."

The **sign** of the eroding depth determines whether the working direction is the direction of the positive coordinate axis (depth +) or of the negative coordinate axis (depth –).

You can enter the eroding depth in **absolute** or **incremental** dimensions.

Miscellaneous function M

You can enter a miscellaneous function in Cycle 17 DISK, such as M36 (eroding ON).

Expansion radius RAD

The TNC feeds the electrode in radial direction (perpendicular to the eroding depth) by the value of the expansion radius.

> The electrode radius Re must be larger than the expansion radius RAD. Otherwise the pocket (disk) will not be completely eroded.

Calculating the expansion radius RAD

If the diameter D of the disk is known, you can calculate the expansion radius RAD from the following data:

- Diameter D of the disk
- Electrode undersize UM
- Electrode minimum undersize UNS
- Electrode radius Re

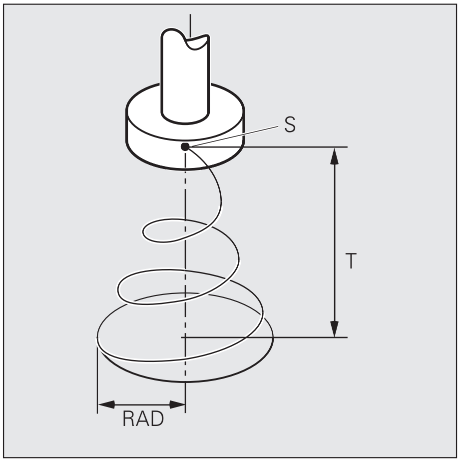
RAD = 0.5 • (UM – UNS) **= 0.5 • D – Re – 0.5 • UNS**

Expansion mode MOD

The expansion mode MOD determines the movement of the electrode while eroding. MOD also influences sparking out and the retraction movement.

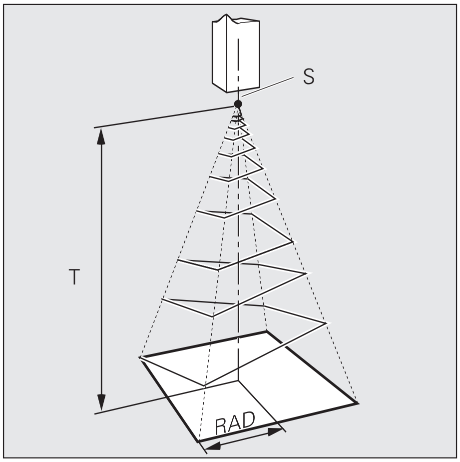
Differences with sparking out

- **Fast sparking-out (MOD = 0 to 3)**
  The TNC ends the cycle when the electrode reaches the final vector V and has eroded one full orbit at the final depth.
- **Complete sparking-out (MOD = 4 to 7)**
  The TNC ends the cycle when the electrode reaches the final vector V and has eroded 1.25 orbits at the final depth.



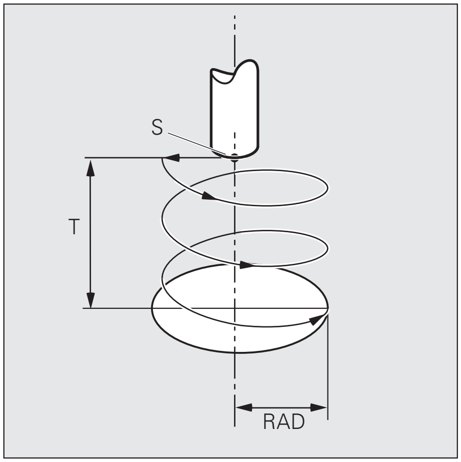
Types of electrode movement

- **Circular expansion (MOD = 0 and 4)**
  From the starting depth S the electrode moves along the surface of a circular cone until it reaches the programmed eroding depth T and the expansion radius RAD (see top illustration).
- **Square expansion (MOD = 1 and 5)**
  From the starting depth S the electrode moves along the surface of a square-base pyramid until it reaches the programmed eroding depth T and the expansion radius RAD (see center illustration).
- **Orbital sinking (MOD = 2 and 6)**
  The electrode moves from the starting point S by the expansion radius RAD in radial direction. It then follows a radial path until reaching the eroding depth. After reaching the eroding depth T, the TNC moves the electrode to the starting point S on a diagonal path.
- **Orbital sinking (MOD = 3 and 7)**
  The electrode moves from the starting point S by the expansion radius RAD in radial direction. It then follows a radial path until reaching the eroding depth (see bottom illustration). After reaching the eroding depth T, the TNC moves the electrode to the starting point S on a diagonal path.



Overview of expansion modes

Movement	Sparking-out	Mode
Circular expansion	Fast Complete	0 4
Quadratic expansion	Fast Complete	1 5
Orbital sinking	Fast, with diagonal retraction Complete, with vertical retraction	2 6
Orbital sinking	Fast, with diagonal retraction Complete, with vertical retraction	3 7



Feed rates for eroding with Cycle 17 DISK

The **feed rate for rotary motion** is the same as the last-programmed feed rate. It is limited by user parameters MP1092 to MP1097.

The **feed rate in the tool axis direction** is determined by the gap control.

Standard behavior with short circuit

In the event of a short circuit, the electrode is stopped and retracted along the infeed vector.
Once the short circuit is eliminated, the TNC moves the electrode back along the infeed vector toward the workpiece but stops a certain distance before the point where the short circuit occurred (this distance is defined in parameter MP2050).

> The machine tool builder may have specified a different retraction behavior in the event of short circuiting than is described here.
> Refer to your machine tool manual.

Cycle 2 ERO.TIME LIM.

Cycle 2 ERO.TIME LIM. (Erosion Time Limit) defines the duration of eroding for:

- Cycle 16 ORBIT
- Cycle 17 ORBIT
- Miscellaneous function M93

During eroding, the TNC interrupts machining when the programmed eroding time is reached.

Enter the eroding time T in minutes in Cycle 2 ERO.TIME LIM.

- Within the program, Cycle 2 ERO.TIME LIM must be located before Cycle 17 DISK or Cycle 16 ORBIT, or before the positioning block with M93.
- Cycle 2 ERO.TIME LIM. influences Q parameter Q153.

To enter Cycle 2 ERO.TIME LIM.

CYCL DEF — Open the cycle directory.

`CYCL DEF 1 GENERATOR`

↓ ENT — Select Cycle 2.0 ERO.TIME LIM.

`ERODING-TIME (MIN) ?`

15 ENT — Enter eroding time T, e.g. T = 15 min.

Example NC blocks

```
2.0  CYCL DEF   ERO.TIME LIM.
2.1  CYCL DEF   T=15
```

Cycle 4 SPARK-OUT TIME

The SPARK-OUT TIME cycle determines how long sparking-out should last.

The defined spark-out time remains effective until you enter a new Cycle 4, or a new program is selected in a Program Run mode. Then the spark-out time set in MP2110 is once again effective.

To enter Cycle 4 SPARK-OUT TIME

CYCL DEF — Open the cycle directory.

CYCL DEF 1 GENERATOR

↓ ENT — Select Cycle 4.0 SPARK-OUT TIME.

SPARKING-OUT TIME IN SECS. ?

5 ENT — Enter the spark-out time T in seconds (for example,T = 5 seconds).

Example NC blocks

```
4.0  CYCL DEF   SPARK-OUT TIME
4.1  CYCL DEF   T=5
```

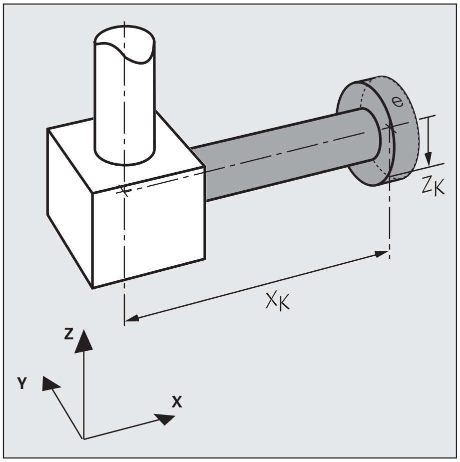
Example for Cycle 3 TOOL DEF

A cavity is to be eroded with the electrode in the drawing at right.

Coordinates of the cavity X = Y = 50 mm
Depth of the cavity Z = – 5 mm

Tool compensation for X_k = – 10 mm
Z_k = + 5 mm

The TNC automatically takes account of the compensation values in the program. You only have to enter the actual coordinates for the position of the cavity, and the eroding depth.



Program section:

.	
.	
.	
`11 CYCL DEF 3.1 TOOL DEF`	Cycle 3 TOOL DEF
`12 CYCL DEF 3.1 T1 R+0`	Tool number, tool radius
`13 CYCL DEF 3.2  X–10 Z+5`	Compensation values
`14 TOOL CALL 1 Z U+0.1`	Tool call
`15 L X+50 Y+50 Z+2`	Pre-position
`16 L Z–5 M36`	Eroding
.	
.	
.	

Example for Cycle 14 CONTOUR GEOMETRY

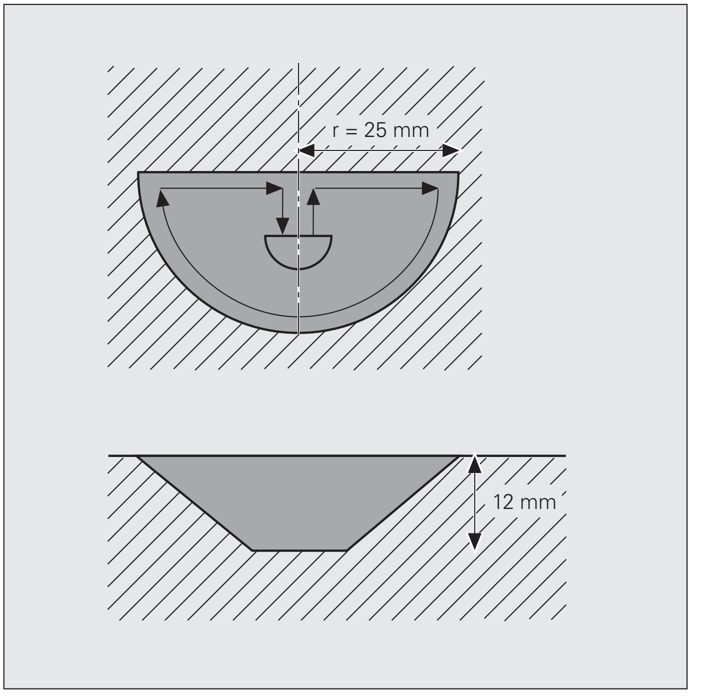
The program GEOMETR describes the geometry of the contour.

The program is called through Cycle 14 CONTOUR GEOMETRY.

The form electrode moves into the material step-by-step according to counting parameter Q5.

The scaling factor is decreased after each infeed, resulting in the diagonal side wall.

Machine parameter **MP7410=1**, meaning the scaling factor does not apply to the Z axis.



Main program:

`0 BEGIN PGM POCKET MM`	Start of program
`1 BLK FORM 0.1 Z X-50 Y-50 Z-30`	Define the workpiece blank
`2 BLK FORM 0.2 X+50 Y+50 Z+0`	
`3 CYCL DEF 1.0 GENERATOR`	Cycle GENERATOR (see "Cycle 1 GENERATOR" on page 133)
`4 CYCL DEF 1.1 P-TAB HDH700`	Select erosion table (here, table HDH700)
`5 CYCL DEF 1.2 MAX=13 MIN=13`	Select power stage 13
`6 TOOL DEF 1 L+0 R+3`	Define the tool
`7 TOOL CALL 1 Z U+0`	Tool call
`8 L Z+50 C+0 R F M37`	Set-up clearance, orient electrode, eroding OFF
`9 L X+0 Y+0 Z+1 R F M`	Pre-positioning
`10 FN 0: Q5 = +8`	Counting parameter
`11 FN 0: Q1 = +1`	Scaling factor
`12 FN 0: Q10= +25`	Contour radius (semicircle)
`13 FN 4: Q12= +Q10 DIV +2`	Auxiliary parameters for pre-positioning in Y direction
`14 FN 0: Q4 = +80`	Parameter for spark-out distance in percent

15 FN 16: Q11 = Q200(Q99)	The diametrical gap according to the current power stage is
	assigned to Q11 (see ”Indexed assignment” on page 198).
16 FN3 Q11 = Q11 * 0.8	Calculation of the vertical gap
17 L Z+Q11 R0 F M36	Pre-positioning with vertical gap; eroding ON
18 LBL 1	Label number
19 CYCL DEF 11.0 SCALING	SCALING cycle
20 CYCL DEF 11.1 SCL Q1	(see ”SCALING FACTOR (Cycle 11)” on page 160)
21 L IY+Q12 R F M	Pre-positioning
22 CYCL DEF 14.0 CONTOUR GEOMETRY	Cycle 14 Contour Geometry (see ”Cycle 14 CONTOUR GEOMETRY” on page 137)
23 CYCL DEF 14.1 IZ-1.5 M36	Incremental eroding depth, eroding ON
24 CYCL DEF 14.2 PGM GEOMETR	Name of the contour program
25 CYCL DEF 14.3 PRC=Q4	Spark-out distance in percent
26 L IY-Q12 R F M37	Retract, eroding OFF
27 FN 2: Q1 = +Q1 - +0.1	New scaling factor
28 FN 2: Q5 = +Q5 - +1	Decrease counter
29 IF +Q5 NE +0 GOTO LBL 1	Jump to LBL1 if counter does not equal zero
30 L Z+50 R0 FMAX M37	Set-up clearance, eroding OFF
31 END PGM POCKET MM	End of program

Contour program:

0 BEGIN PGM GEOMETR MM	
1 CC IX+0 IY+0	Current position as center of circle
2 FN 3: Q11= +Q10 * +2	Calculate diameter
3 L IX+Q10 IY+0 R F M	Traverse contour (block 3 to 5)
4 C IX-Q11 IY+0 DR- R F M	
5 L IX+Q10 R F M	
6 END PGM GEOMETR MM	

Practice examples: Eroding with Cycle 16 ORBIT

Workpiece geometry

Cavity diameter D = 24 mm
Eroding depth T = –10 mm

Electrode data

Cylindrical electrode
Electrode radius Re = 9.9 mm
Electrode undersize U = 4.2 mm
Determining the eroding gap B through indexed assignment

Calculation of the expansion radius

Expansion radius for Cycle 16 ORBIT

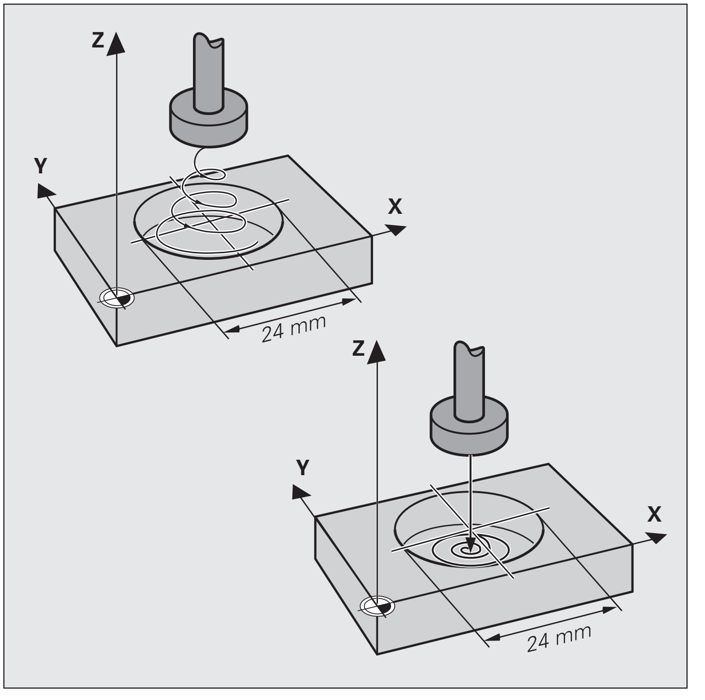
RAD = 0.5 • (UM – UNS)
RAD = 0.5 • D – Re – 0.5 • UNS

Example 1, top illustration:

Pre-position over the workpiece surface, circular expansion.

Example 2, bottom illustration:

Erode to –10 mm depth, circular expansion without pecking.



Cycle 16 ORBIT in the part program, example 1

0 BEGIN PGM EX1 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z–20	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 CYCL DEF 1.0 GENERATOR	Cycle GENERATOR (see "Cycle 1 GENERATOR" on page 133)
4 CYCL DEF 1.1 P-TAB CUST1	Desired erosion table
5 CYCL DEF 1.2 MAX=10 MIN=5	Maximum power stage = 10, minimum power stage = 5
6 TOOL DEF 1 L+0 R+9.9	Electrode radius
7 TOOL CALL 1 Z U+4.2	Undersize
8 L Z+50 C+0 R0 F MAX M37	Pre-position to set-up clearance, eroding OFF
9 L X+50 Y+50 Z+1 R F MAX	Pre-position over the workpiece surface
10 FN 0: Q1 = +11	Assign incremental depth to Q1
11 LBL1	Label number
12 FN16: Q10 = Q200(Q99)	The diametrical gap according to the current power stage is
	assigned to Q10 (see "Indexed assignment" on page 198)
13 FN2: Q9 = +Q158 - +Q10	Electrode undersize UM minus electrode undersize UNS

14 FN4: Q8 = +Q9 DIV +2	Calculation of the expansion radius RAD
15 FN3: Q7 = +Q10 * +0.8	Calculation of the vertical gap
16 FN2: Q6 = +Q1 - +Q7	Decrease incremental depth by the vertical gap
17 CYCL DEF 16.0 ORBIT	Cycle ORBIT (see "Cycle 16 ORBIT" on page 139)
18 CYCL DEF 16.1 IZ-Q6 M36	Incremental eroding depth IZ=–Q6, eroding ON
19 CYCL DEF 16.2 RAD=Q8 DIR=0	Expansion radius RAD=Q8, erosion movement
	counterclockwise DIR=0
20 CYCL DEF 16.3 PAT=0 SPO=0	Circular expansion PAT=0, spark-out mode SPO=0
21 IF +Q99 EQU +Q151 GOTO LBL 99	Inquiry if minimum power stage has been reached
22 FN 2: Q99= +Q99 - +1	Decrease current power stage by 1
23 FN 9: IF +0 EQU +0 GOTO LBL 1	Jump to LBL1, machine again with lower power stage
24 LBL 99	LBL 99 is reached when machining with the lowest
	power stage is completed
25 L Z+50 R0 F MAX M37	Retract to set-up clearance, eroding OFF
26 END PGM EXORB1 MM	

Cycle 16 ORBIT in the part program, example 2

0 BEGIN PGM EX2 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 CYCL DEF 1.0 GENERATOR	Cycle GENERATOR (see "Cycle 1 GENERATOR" on page 133)
4 CYCL DEF 1.1 P-TAB CUST1	Desired erosion table
5 CYCL DEF 1.2 MAX=10 MIN=5	Maximum power stage = 10, minimum power stage = 5
6 TOOL DEF 1 L+0 R+9.9	Electrode radius
7 TOOL CALL 1 Z U+4.2	Undersize
8 L Z+50 C+0 R0 F MAX M37	Pre-position to setup clearance, eroding OFF
9 L X+50 Y+50 Z+1 R F MAX	Pre-position over the workpiece surface
10 FN 0: Q1 = +11	Assign incremental depth to Q1
11 LBL1	Label number
12 FN16: Q10 = Q200(Q99)	The diametrical gap according to the current power stage is
	assigned to Q10 (see "Indexed assignment" on page 198)
13 FN2: Q9 = +Q158 - +Q10	Electrode undersize UM minus electrode undersize UNS
14 FN4: Q8 = +Q9 DIV +2	Calculation of the expansion radius RAD
15 FN3: Q7 = +Q10 * +0.8	Calculation of the vertical gap
16 FN2: Q6 = +Q1 - +Q7	Decrease incremental depth by the vertical gap
17 L IZ - +Q6 R0 F M36	Erode to end depth, eroding ON
18 CYCL DEF 16.0 ORBIT	Cycle ORBIT (see "Cycle 16 ORBIT" on page 139)
19 CYCL DEF 16.1 IZ+0 M36	Erode to end depth, eroding ON

`20 CYCL DEF 16.2 RAD=Q8 DIR=0`	Expansion radius RAD=Q8, erosion movement
	counterclockwise DIR=0
`21 CYCL DEF 16.3 PAT=0 SPO=0`	Circular expansion PAT=0, spark-out mode SPO=0
`22 IF +Q99 EQU +Q151 GOTO LBL 99`	Inquiry if minimum power stage has been reached
`23 FN 2: Q99= +Q99 - +1`	Decrease current power stage by 1
`24 FN 9: IF +0 EQU +0 GOTO LBL 1`	Jump to LBL1, machine again with lower power stage
`25 LBL 99`	LBL 99 is reached when machining with the lowest
	power stage is completed
`26 L Z+50 R0 F MAX M37`	Retract to set-up clearance, eroding OFF
`27 END PGM EX2 MM`	

Practice examples: Eroding with Cycle 17 DISK

Workpiece geometry

Cavity diameter D = 24 mm
Eroding depth T = –10 mm

Electrode data

Cylindrical electrode
Electrode radius Re = 9.9 mm
Electrode undersize U = 4.2 mm
Width of the erosion gap B= 0.1 mm

Calculation of the expansion radius

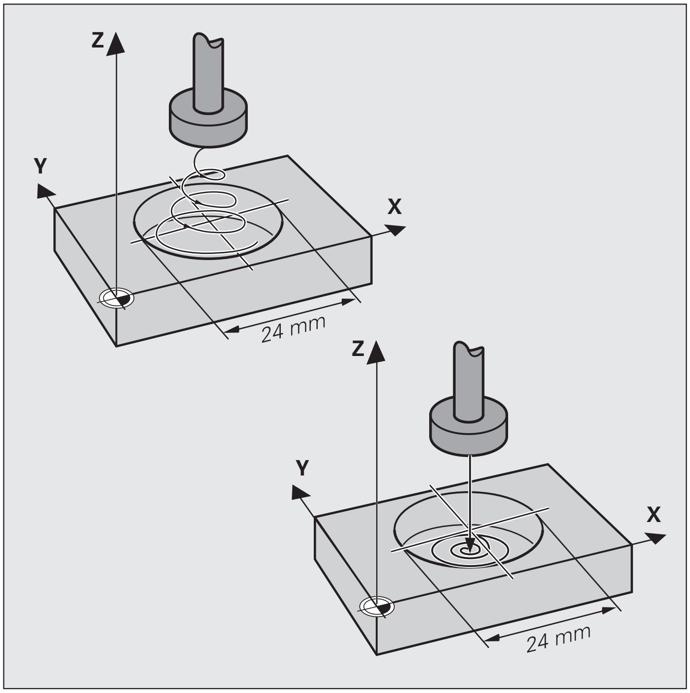
Expansion radius for Cycle 17 DISK
RAD = (0.5 • 4.2 mm) – 0.1 mm = 2 mm

Example 1, top illustration:

Pre-position over the workpiece surface, circular expansion.

Example 2, bottom illustration:

Erode to –10 mm depth, circular expansion without pecking.



Cycle 17 DISK in the part program, example 1

`0 BEGIN PGM EXDISK1 MM`	
`1 BLK FORM 0.1 Z X+0 Y+0 Z-20`	
`2 BLK FORM 0.2 X+100 Y+100 Z+0`	
`3 CYCL DEF 1.0 GENERATOR`	Cycle GENERATOR (see "Cycle 1 GENERATOR" on page 133)
`4 CYCL DEF 1.1 P-TAB CUST1`	Desired erosion table
`5 CYCL DEF 1.2 MAX=8 MIN=8`	Select power stage
`6 TOOL DEF 1 L+0 R+9.9`	Electrode length, electrode radius
`7 TOOL CALL 1 Z U+4.2`	Undersize
`8 L X+50 Y+50 Z+1 R0 F MAX`	Pre-positioning
`9 CYCL DEF 17.0 DISK`	Cycle 17 DISK (see "Cycle 17 DISK" on page 142)
`10 CYCL DEF 17.1 Z-10 R F M36`	Eroding depth Z = –10 mm, eroding ON
`11 CYCL DEF 17.2 RAD=2 MOD=0`	Expansion radius RAD = 2 mm, circular expansion
`13 L Z+100 R F MAX M37`	Retract to set-up clearance, eroding OFF
`14 END PGM EXDISK1 MM`	

Cycle 17 DISK in the part program, example 2

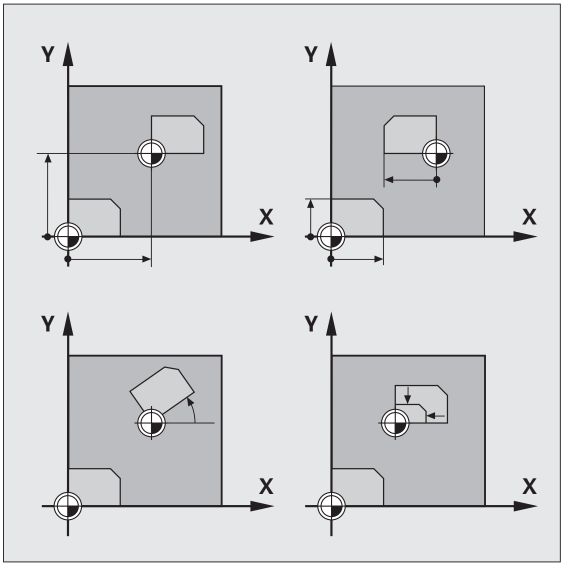
`0 BEGIN PGM EXDISK2 MM`	
`1 BLK FORM 0.1 Z X+0 Y+0 Z-20`	
`2 BLK FORM 0.2 X+100 Y+100 Z+0`	
`3 CYCL DEF 1.0 GENERATOR`	Cycle GENERATOR (see "Cycle 1 GENERATOR" on page 133)
`4 CYCL DEF 1.1 P-TAB 20`	Desired erosion table
`5 CYCL DEF 1.2 MAX=8 MIN=8`	Select power stage
`6 TOOL DEF 1 L+0 R+9.9`	Electrode length, electrode radius
`7 TOOL CALL 1 Z U+4.2`	Undersize
`8 L X+50 Y+50 Z+1 R0 F MAX M`	Pre-position over the workpiece surface
`9 L Z-10 R F M36`	Erode to end depth, eroding ON
`9 CYCL DEF 17.0 DISK`	Cycle 17 DISK
`10 CYCL DEF 17.1 IZ+0 M36`	Incremental eroding depth, eroding ON
`11 CYCL DEF 17.2 RAD=2 MOD=0`	Expansion radius RAD= 2 mm, circular expansion
`13 L Z+100 R F MAX M37`	Retract to set-up clearance, eroding OFF
`14 END PGM EXDISK2 MM`	

8.5 Coordinate Transformation Cycles

Cycles for electrode definition

You can enter electrode data in this cycle in a manner similar to the NC function TOOL DEF.
In addition, you can program an electrode compensation in up to four axes.

Coordinate transformation cycles



Once a contour has been programmed, you can position it on the workpiece at various locations and in different sizes through the use of coordinate transformations.

For example you can

- Move a contour: DATUM SHIFT (Cycle 7)
- Mirror a contour: MIRROR IMAGE (Cycle 8)
- Rotate a contour: ROTATION (Cycle 10)
- Reduce or increase the size of a contour: SCALING FACTOR (Cycle 11)

The original contour must be marked in the main part program as a subprogram or program section repeat.

In addition, the function "Tilt working plane" can be used to execute Cycle 16 ORBIT, Cycle 17 DISK or an OEM cycle in a tilted system of coordinates.

Canceling coordinate transformations

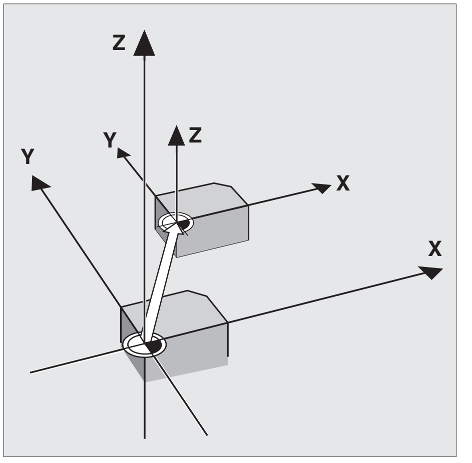
You can cancel a coordinate transformation in the following ways:

- Define cycles for basic behavior with a new value, such as scaling factor 1.0
- Execute the miscellaneous function M02 or M30, or an END PGM block (depending on machine parameters)
- Select a new program

DATUM SHIFT (Cycle 7)

Application

Machining operations can be repeated at various locations on the workpiece with a datum shift.



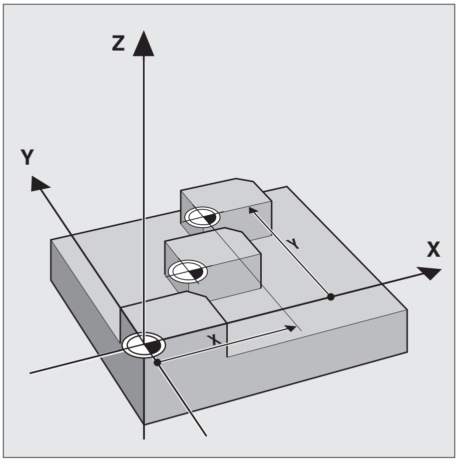
Effect

When the DATUM shift cycle is defined, all coordinate data is based on the new datum.
The datum shift is indicated in the status display with the index T by the shifted axes.

Input

Enter the coordinates of the new datum (zero point) for up to 5 axes. Absolute values are referenced to the zero point, which is determined by the manual datum setting. Incremental values are referenced to the datum which was last valid (this may be a datum which has already been shifted).

If you are working with the datum table, enter the name of the datum (with the # key) from the table and the name of the datum table from which the TNC is to activate the datum shift. If you do not enter a name, the TNC automatically uses the datum table 0.D. A selected datum table remains active until you activate another table at a later stage in the program.

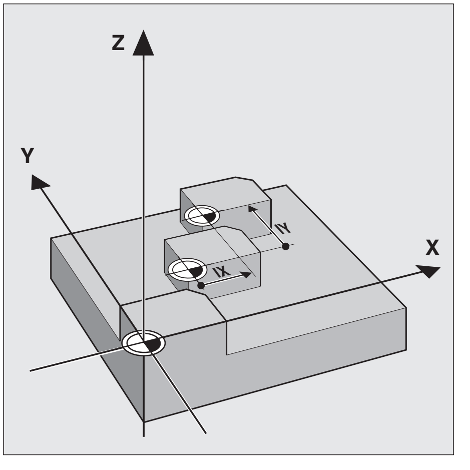


The status display STATUS COORD. TRANSF. shows you the datum table and the datum number that are currently active.

Cancellation

A datum shift is canceled by entering the datum shift coordinates 0, or with the number #0.

If you combine coordinate transformations, note that the datum shift must be programmed before other transformations.



Working with datum tables

The TNC can store several datum tables. Depending on the configuration of your machine tool, a new datum table includes four or five axes.

Editing a datum table:

- Press the PGM NAME or PGM MGT key in the PROGRAMMING AND EDITING mode of operation.
- Enter the name of the datum table.
  The selected datum table appears on the screen. You can store the coordinates for up to 999 datum points in this table. If necessary, you can enlarge the table with the INSERT soft key, and enter the desired datum number in column D.

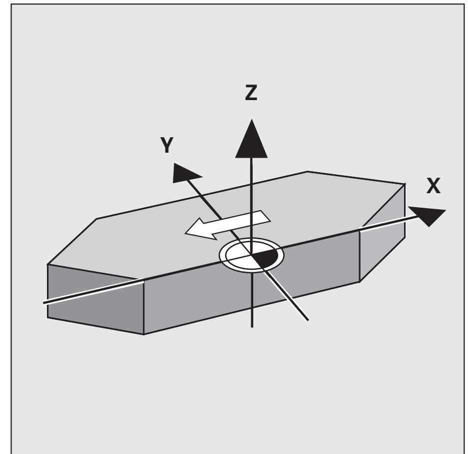
The TNC writes the datum number and coordinates in the Q parameters Q80 to Q85.

With the miscellaneous functions M38 and M39 you can write coordinates to and from the active datum table. M38 and M39 allow you to store any positions as datum points in the table 0.D (see also "Q parameters for the datum table: Q81 to Q84" on page 206).

Depending on the setting of user parameter 7411, a datum shift in the fourth axis will also result in a rotation (see also "Selecting the General User Parameters" on page 246).

If the tool axis is not the Z axis, C from the datum table will only result in a shift, not a rotation.

MIRROR IMAGE (Cycle 8)



Function

The TNC can machine the mirror image of a contour in the working plane.

Input

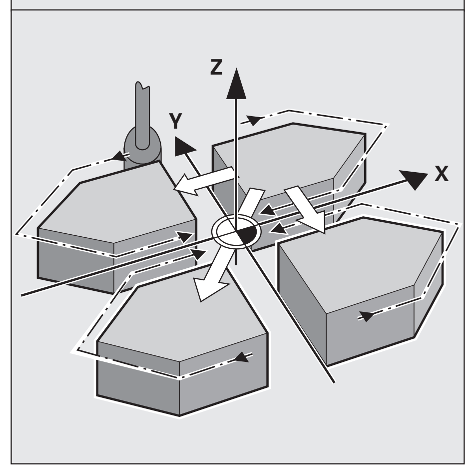
Enter the axis that you wish to mirror.
The tool axis cannot be mirrored.

Cancellation

The cycle is canceled by replying with NO ENT to the dialog question.

Effect

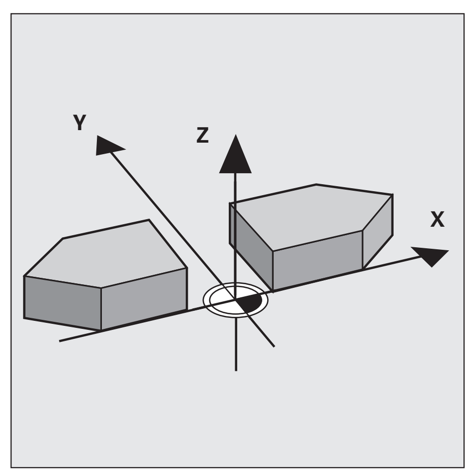
The mirror image cycle becomes effective as soon as it is defined in the program. The mirrored axis is indicated in the status display with the index S by the mirrored axes.



- When one axis is mirrored, the machining direction of the electrode is reversed.
- If **two** axes are mirrored, the machining direction remains the same.

The mirror image depends on the location of the datum:

- If the datum lies **on** the contour to be mirrored:
  The part simply ”flips over” (see top illustration).
- If the datum lies **outside** the contour to be mirrored:
  The part also ”jumps” to another location (see bottom illustration).



ROTATION (Cycle 10)

Function

The coordinate system can be rotated about the active datum in the working plane within a program.

Effect

The rotation takes effect as soon as it is defined in the program. Cycle 10 ROTATION cancels radius compensation RR/RL.

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 shown in the status display (ROT).

Definition of the plane of rotation

When the ROTATION cycle is activated for the first time, the plane of rotation is perpendicular to the tool axis defined in the tool call block.

If later a TOOL CALL block with a different tool axis is executed, the plane of rotation will not change.

Input

Enter the rotation angle in degrees (°).
Input range: –360° to +360° (absolute or incremental).

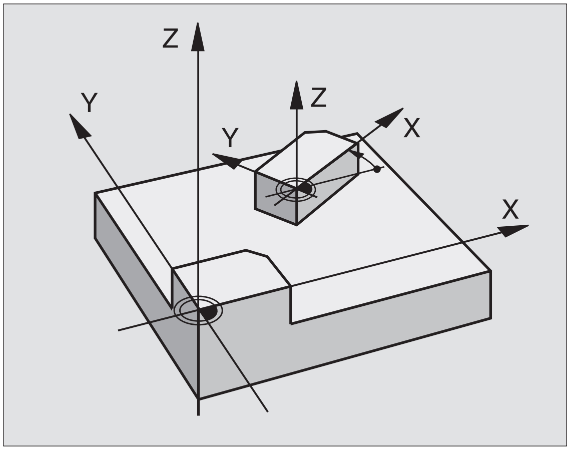
Effect on Q parameters

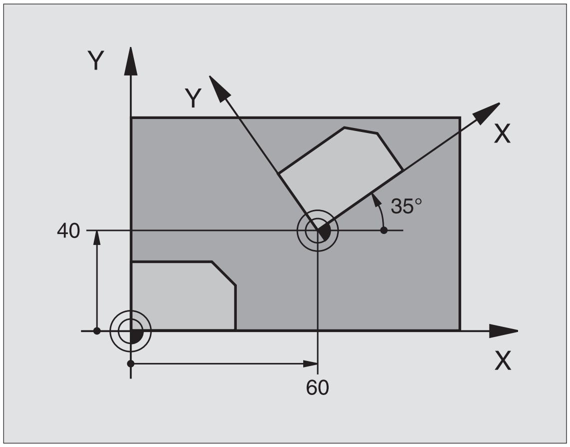
The plane of rotation influences Q parameter Q112:

- X/Y plane Q112 = 2
- Y/Z plane Q112 = 0
- Z/X plane Q112 = 1
- No plane defined Q112 = –1

Cancellation

A rotation is canceled by entering a rotation angle of 0°.





Example: NC blocks

```
12 CALL LBL1
13 CYCL DEF 7.0 DATUM SHIFT
14 CYCL DEF 7.1 X+60
15 CYCL DEF 7.2 Y+40
16 CYCL DEF 10.0 DREHUNG
17 CYCL DEF 10.1 ROT+35
18 CALL LBL1
```

SCALING FACTOR (Cycle 11)

Function

The scaling factor cycle allows contours to be enlarged or reduced in size within a program, enabling you to program shrinkage and oversize allowances.

Effect

The scaling factor cycle takes effect as soon as it is defined. The scaling factor can be applied

- in the working plane, or on all three coordinate axes at the same time (depending on MP7410)
- to the dimensions in cycles
- to the parallel axes U,V,W

The scaling factor is shown in the status display under SCL.

Input

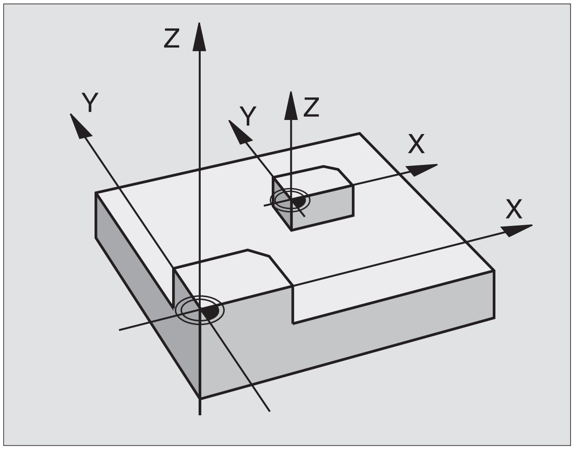
The cycle is defined by entering the factor SCL. The TNC multiplies the coordinates and radii by the SCL factor (as described under "Effect" above).
Enlargement: SCL greater than 1 (up to 99.999 999)
Reduction: SCL less than 1 (down to 0.000 001)

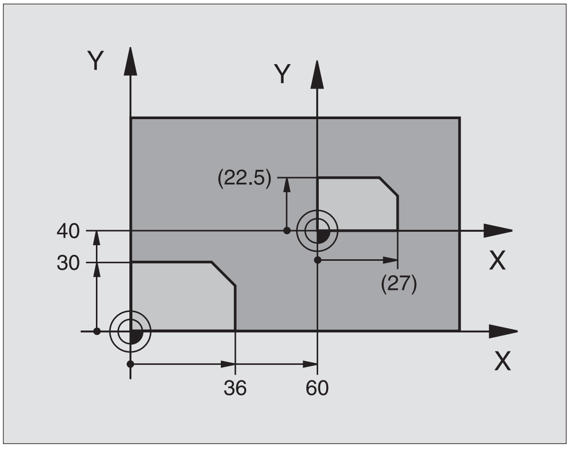
Cancellation

To cancel the scaling factor, enter a scaling factor of 1.

Prerequisite

It is advisable to set the datum to an edge or a corner of the contour before enlarging or reducing the contour.

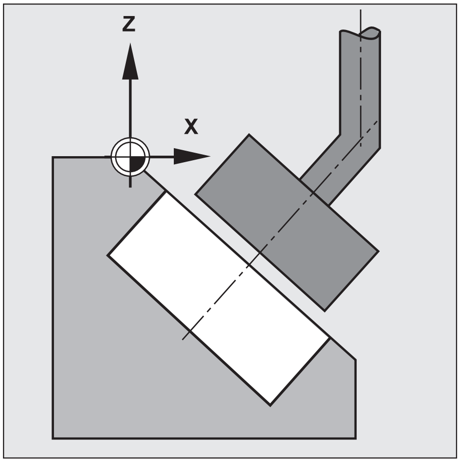




Example: NC blocks

```
11 CALL LBL1
12 CYCL DEF 7.0 DATUM SHIFT
13 CYCL DEF 7.1 X+60
14 CYCL DEF 7.2 Y+40
15 CYCL DEF 11.0 SCALING
16 CYCL DEF 11.1 SCL 0.75
17 CALL LBL1
```

WORKING PLANE (Cycle 19)



Function

With Cycle 19, it is possible to tilt linear traverse and machining with Cycle 16 ORBIT, Cycle 17 DISK or an OEM cycle at random in a 3-D plane. Thus, execution of inclined eroding-cycles can be made simple.

Effect

After a cycle definition WORKING PLANE, the TNC tilts the subsequent machining blocks around the datum which was last set in the MANUAL mode (active datum).

Input

You enter:

- Tilt angle A, corresponding to the rotation about the X axis. This can be programmed with the orange key X.
- Tilt angle B, corresponding to the rotation about the Y axis. This can be programmed with the orange key Y.
- Tilt angle C, corresponding to the rotation about the Z axis. This can be programmed with the orange key Z.

The TNC displays the current active tilt angles in the STATUS TILT display.

Input range: –360° to +360° (only absolute values possible).

Cancellation

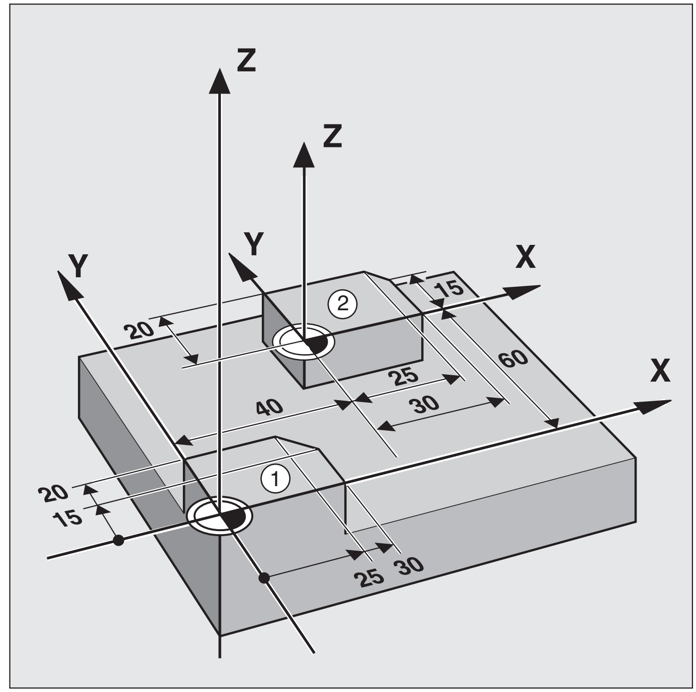
To cancel the tilt angle, redefine the WORKING PLANE cycle and enter an angular value of 0° for all axes of rotation, or select a new program.

- Coordinate transformations, e.g. a datum shift, are also effective when the Tilt working plane function is active.
- An active basic rotation is calculated in the same way as a tilting of the machine plane about the C axis.
- When creating OEM cycles, remember that traverse paths within the cycle may only be programmed with L blocks.

Practice example: Datum shifting

A machining sequence in the form of a subprogram is to be executed twice:

- once, reference to the specified datum 1 X+0/Y+0, and
- a second time, reference to the shifted datum 2 X+40/Y+60.



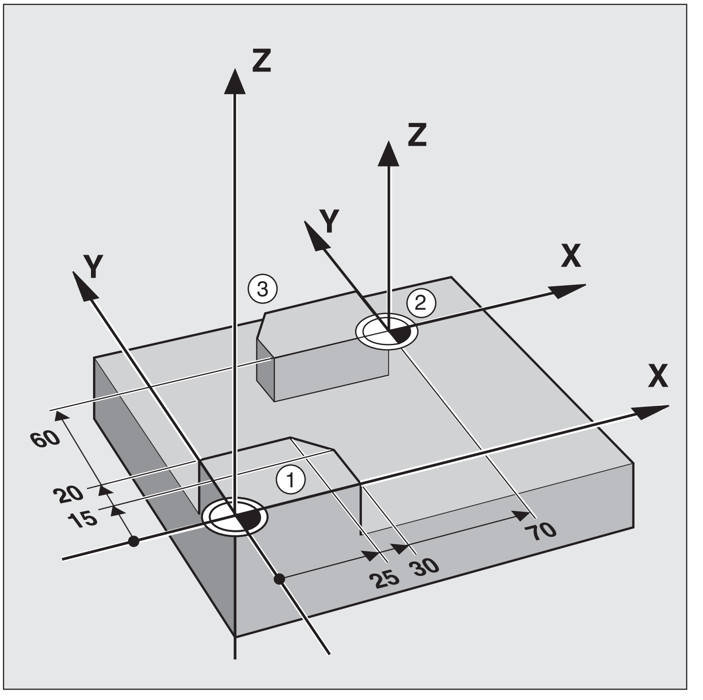
DATUM SHIFT cycle in a part program:

`0 BEGIN PGM DATUM MM`	
`1 BLK FORM 0.1 Z X+0 Y+0 Z-20`	
`2 BLK FORM 0.2 X+100 Y+100 Z+0`	
`3 CYCL DEF 1.0 GENERATOR`	Cycle GENERATOR (see "Cycle 1 GENERATOR" on page 133)
`4 CYCL DEF 1.1 P-TAB HDH700`	Select erosion table (here, table HDH700)
`5 CYCL DEF 1.2 MAX=5 MIN=5`	Select power stage
`6 TOOL DEF 1 L+0 R+4`	Define the tool
`7 TOOL CALL 1 Z U+0.05`	Tool call
`8 L Z+100 R0 F MAX M`	
`9 CALL LBL 1`	Without datum shift
`10 CYCL DEF 7.0 DATUM SHIFT`	Datum shift in the X/Y plane
`11 CYCL DEF 7.1 X+40`	
`12 CYCL DEF 7.2 Y+60`	
`13 CALL LBL 1`	With datum shift
`14 CYCL DEF 7.0 DATUM SHIFT`	Reset the datum shift
`15 CYCL DEF 7.1 X+0`	

16 CYCL DEF 7.2 Y+0	
17 L Z+100 R0 F MAX M2	End of main program
18 LBL 1	Start of the subprogram for the geometry of the original contour
19 L X–10 Y–10 R0 F MAX M	Pre-positioning in the X/Y plane
20 L Z+2 R FMAX M	Pre-positioning in the Z plane
21 L Z–5 R F M36	Move to end depth, eroding ON
22 L X+0 Y+0 RL F M	Traverse the first contour point
23 L Y+20 R F M	
24 L X+25 R F M	
25 L X+30 Y+15 R F M	
26 L Y+0 R F M	
27 L X+0 R F M	
28 L X–10 Y–10 R0 F MAX M37	Retract in the X/Y plane, eroding OFF
29 L Z+2 R F MAX M	Retract in Z direction
30 LBL 0	End of subprogram
31 END PGM DATUM MM	

Practice 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 once mirrored in X3 at position X+70/Y+602.



MIRROR IMAGE cycle in a part program:

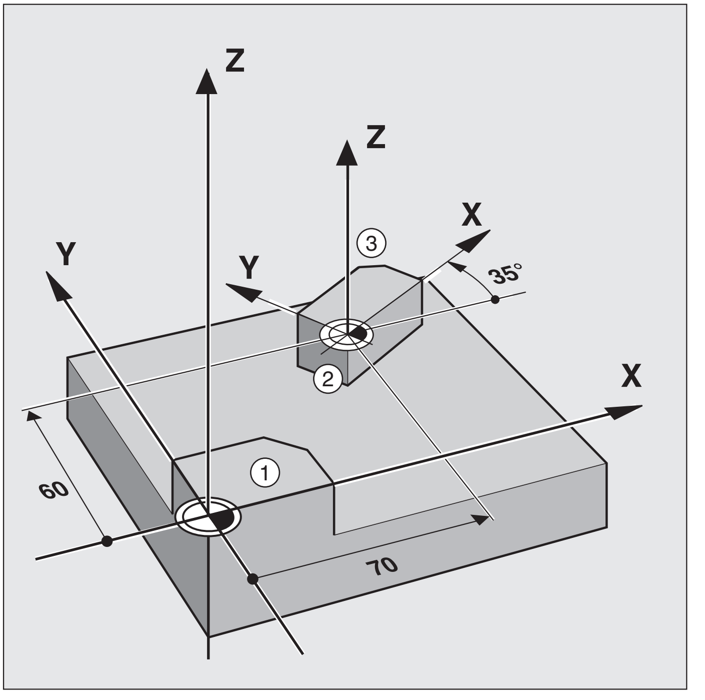
`0 PGM MIRROR MM`	
`1 BLK FORM 0.1 Z X+0 Y+0 Z-20`	
`2 BLK FORM 0.2 X+100 Y+100 Z+0`	
`3 CYCL DEF 1.0 GENERATOR`	Cycle GENERATOR (see "Cycle 1 GENERATOR" on page 133)
`4 CYCL DEF 1.1 P-TAB CUST1`	Select erosion table (here, CUST1)
`5 CYCL DEF 1.2 MAX=12 MIN=12`	Select power stage
`6 TOOL DEF 1 L+0 R+3`	Define the tool
`7 TOOL CALL 1 Z U+0`	Tool call
`8 L Z+100 R0 F MAX M`	
`9 CALL LBL 1`	Unmirrored 1; mirrored version
`10 CYCL DEF 7.0 DATUM SHIFT`	1. Datum shift 2
`11 CYCL DEF 7.1 X+70`	
`12 CYCL DEF 7.2 Y+60`	
`13 CYCL DEF 8.0 MIRROR IMAGE`	2. Mirror image 3
`14 CYCL DEF 8.1 X`	
`15 CALL LBL 1`	3. Subprogram call

16 CYCL DEF 8.0 MIRROR IMAGE	Cancel mirror image
17 CYCL DEF 8.1	
18 CYCL DEF 7.0 DATUM SHIFT	Reset the datum shift
19 CYCL DEF 7.1 X+0	
20 CYCL DEF 7.2 Y+0	
21 L Z+100 R0 F MAX M2	End of main program
22 LBL 1	Start of the subprogram for the geometry of the original contour
23 L X-10 Y-10 R0 F MAX M	Pre-positioning in the X/Y plane
24 L Z+2 R F MAX M	
25 L Z-5 R F M36	Move to end depth, eroding ON
26 L X+0 Y+0 RL F M	Traverse the first contour point
27 L Y+20 R F M	
28 L X+25 R F M	
29 L X+30 Y+15 R F M	
30 L Y+0 R F M	
31 L X+0 R F M	
32 L X–10 Y–10 R0 F MAX M37	Retract in the X/Y plane, eroding OFF
33 L Z+2 R F MAX M	
34 LBL 0	End of subprogram
35 END PGM MIRROR MM	

Practice example: Rotation

A contour section (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.

If the tool axis is parallel to axis IV (for example, Z and C), the ROTATION cycle will cause a shift in axis IV by the same angle as is programmed in the ROTATION cycle.



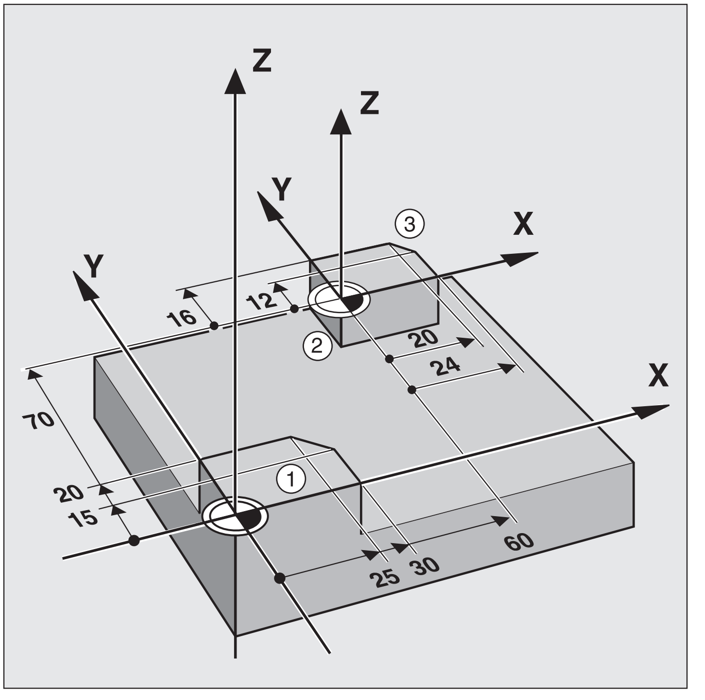
DATUM SHIFT cycle in a part program:

`0 BEGIN PGM ROTAT MM`	
`1 BLK FORM 0.1 Z X+0 Y+0 Z-20`	
`2 BLK FORM 0.2 X+100 Y+100 Z+0`	
`3 CYCL DEF 1.0 GENERATOR`	Cycle GENERATOR (see "Cycle 1 GENERATOR" on page 133)
`4 CYCL DEF 1.1 P-TAB 75`	Desired erosion table
`5 CYCL DEF 1.2 MAX=7 MIN=7`	Select power stage
`6 TOOL DEF 1 L+0 R+5,5`	
`7 TOOL CALL 1 Z U+1`	
`8 L Z+100 R0 F MAX M`	
`9 CALL LBL 1`	Unrotated version 1
`10 CYCL DEF 7.0 DATUM SHIFT`	Rotated version. Sequence:
`11 CYCL DEF 7.1 X+70`	
`12 CYCL DEF 7.2 Y+60`	1. Datum shift 2
`13 CYCL DEF 10.0 ROTATION`	2. Rotate 3
`14 CYCL DEF 10.1 ROT +35`	
`15 CALL LBL 1`	3. Subprogram call

16 CYCL DEF 10.0 ROTATION	Reset the rotation
17 CYCL DEF 10.1 ROT 0	
18 CYCL DEF 7.0 DATUM SHIFT	Cancel datum shift
19 CYCL DEF 7.1 X+0	
20 CYCL DEF 7.2 Y+0	
21 L Z+100 R0 F MAX M2	End of main program
22 LBL 1	Start of the subprogram for the geometry of the original contour
.	
.	
.	
LBL 0	End of subprogram
END PGM ROTAT MM	

Practice example: Scaling factor

A contour section (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:

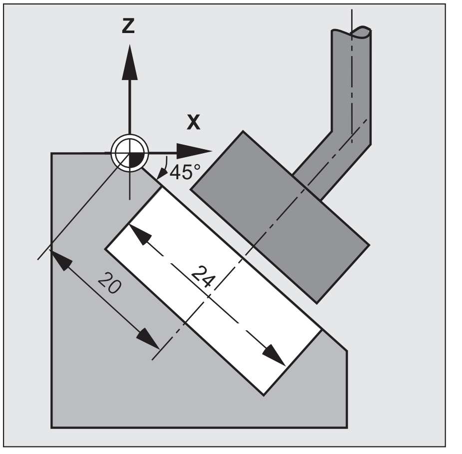
0 BEGIN PGM SCALING MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z–20	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 CYCL DEF 1.0 GENERATOR	Cycle GENERATOR (see "Cycle 1 GENERATOR" on page 133)
4 CYCL DEF 1.1 P-TAB 100	Desired erosion table
5 CYCL DEF 1.2 MAX=7 MIN=7	Select power stage
6 TOOL DEF 1 L+0 R+3	Define the tool
7 TOOL CALL 1 Z U+0.2	Tool call
8 L Z+100 R0 F MAX M	
9 CALL LBL 1	Version in original size 1
10 CYCL DEF 7.0 DATUM SHIFT	Version with scaling factor. Sequence:
11 CYCL DEF 7.1 X+60	
12 CYCL DEF 7.2 Y+70	1. Datum shift 2
13 CYCL DEF 11.0 SCALING	2. Define scaling factor 3
14 CYCL DEF 11.1 SCL 0.8	
15 CALL LBL 1	3. Call subprogram (scaling factor effective)

`16 CYCL DEF 11.0 SCALING`	Cancel transformations
`17 CYCL DEF 11.1 SCL 1`	
`18 CYCL DEF 7.0 DATUM SHIFT`	Reset the datum shift
`19 CYCL DEF 7.1 X+0`	
`20 CYCL DEF 7.2 Y+0`	
`21 L Z+100 R0 F MAX M2`	End of main program
`22 LBL 1`	Start of the subprogram for the geometry of the original contour
`23 L X-10 Y-10 R0 F MAX M`	Pre-positioning in the X/Y plane
`24 L Z+2 R F MAX M`	
`25 L Z-5 R F M36`	Move to end depth, eroding ON
`26 L X+0 Y+0 RL F M`	
`27 L Y+20 R F M`	
`28 L X+25 R F M`	
`29 L X+30 Y+15 R F M`	
`30 L Y+0 R F M`	
`31 L X+0 R F M`	
`32 L X–10 Y–10 R0 F MAX M37`	Retract in the X/Y plane, eroding OFF
`33 L Z+2 R F MAX M`	
`34 LBL 0`	
`35 END PGM SCALING MM`	

Practice examples: Tilt the working plane with Cycle 17 DISK

Execute disk cycle with 45° tilt in the B axis, depth = 10 mm.

For calculation of the cycle parameters, see example Cycle DISK.



WORKING PLANE cycle in a part program:

`0 BEGIN PGM CYC19 MM`	
`1 BLK FORM 0.1 Z X+0 Y+0 Z-40`	
`2 BLK FORM 0.2 X+100 Y+100 Z+0`	
`3 CYCL DEF 1.0 GENERATOR`	Cycle GENERATOR (see "Cycle 1 GENERATOR" on page 133)
`4 CYCL DEF 1.1 P-TAB 100`	Desired erosion table
`5 CYCL DEF 1.2 MAX=7 MIN=7`	Select power stage
`6 TOOL DEF 1 L+0 R+9.9`	Define the tool
`7 TOOL CALL 1 Z U+4.2`	Tool call
`8 L Z+100 R0 F MAX`	Pre-position to set-up clearance
`9 CYCL DEF 19.0 WORKING PLANE`	Define Cycle 19 WORKING PLANE
`10 CYCL DEF 19.1 B+45`	Tilt working plane about the B axis
`11 L X+20 Y+20 R0 F MAX M`	Pre-position to center of disk
`12 L Z+1 R0 F MAX M`	Pre-position over the workpiece surface
`13 CYCL DEF 17.0 DISK`	Define Cycle 17 DISK
`14 CYCL DEF 17.1 Z-10 M36`	Eroding depth Z = –10 mm, eroding ON
`15 CYCL DEF 17.2 RAD=2 MOD=0`	Expansion radius RAD = 2 mm, circular expansion
`16 L Z+50 F MAX M37 M`	Retract diagonally to safety clearance, eroding OFF
`17 CYCL DEF 19.0 WORKING PLANE`	Reset Cycle 19 WORKING PLANE
`18 CYCL DEF 19.1 B+0`	
`19 L R F M2`	
`20 END PGM CYC19 MM`	

8.6 Other Cycles

DWELL TIME (Cycle 9)

Function

This cycle causes the execution of the next block within a running program to be delayed by the programmed dwell time.

Effect

The cycle takes effect as soon as it is defined. Modal conditions are not affected.

Input

Enter the dwell time in seconds.
Input range: 0 to 30 000 seconds (approx. 8.3 hours) in increments of 0.001 seconds.

PGM-CALL (Cycle 12)

Application and effect

Routines that are programmed by the user (such as special eroding cycles, curves or geometrical modules) can be written as main programs and set equal to machining cycles.

These main programs can then be called like fixed cycles.

Input

Enter the name of the program to be called.

Calling Cycle 12 PGM-CALL

The program is called with

- CYCL CALL (separate block) or
- M99 (blockwise) or
- M89 (performed after every positioning block, depending on machine parameters).

Cancellation

You can cancel M89 (cycle call after every block) as follows:

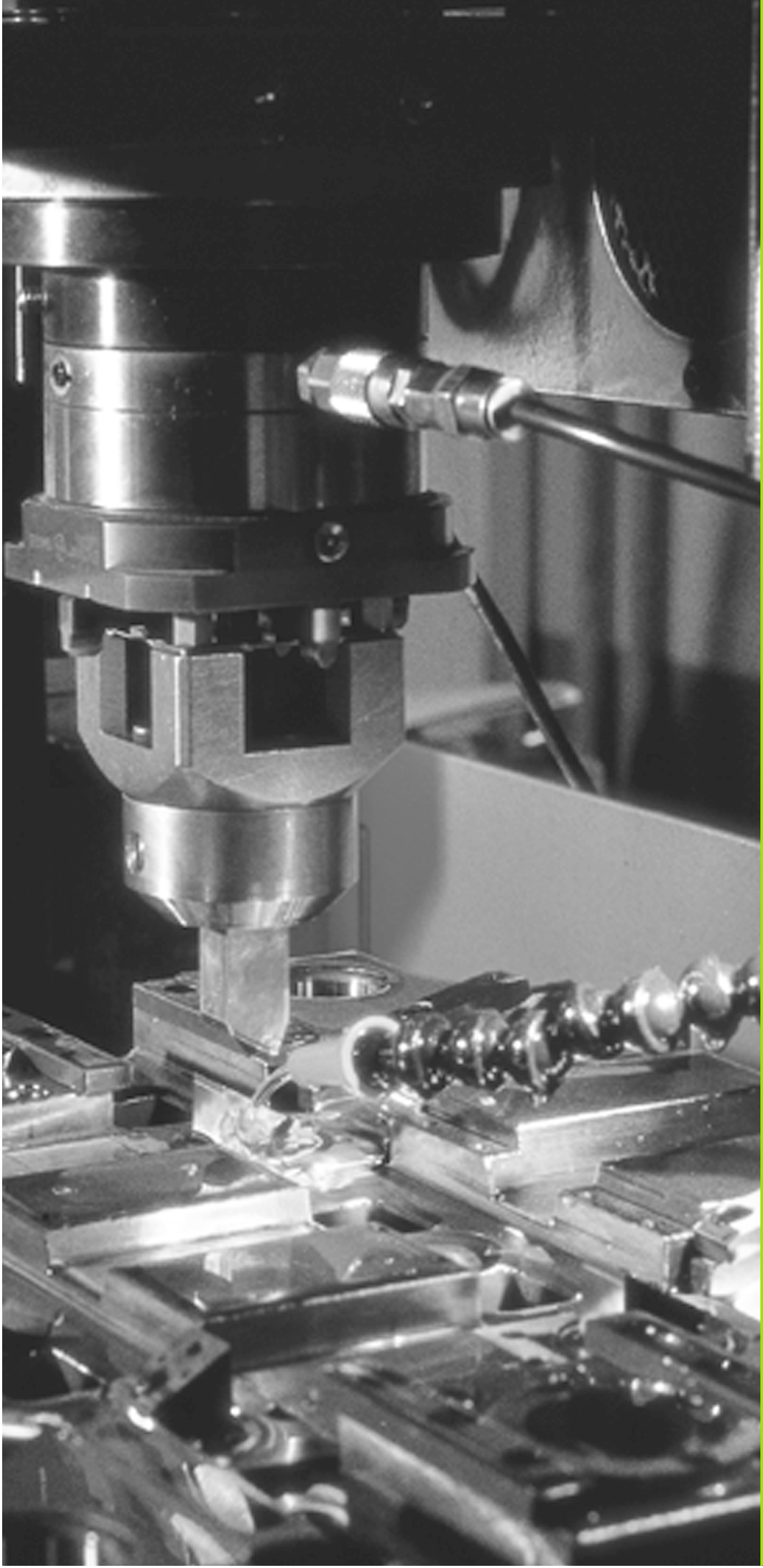
- With M99 (the program is called one more time)
- With CYCL CALL (the program is called one more time)
- By redefining Cycle 12

Example: Program call

A callable program 50 is to be called into a program via a cycle call.

The part program:

`...`	
`...`	
`11 CYCL DEF 12.0 PGM CALL`	Definition
`12 CYCL DEF 12.1 PGM 50`	"Program 50 is a cycle"
`13 L X+20 Y+50 R FMAX M99`	Call program 50
`...`	
`...`	



9

Programming: Subprograms and Program Section Repeats

9.1 Labeling 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

The beginnings of subprograms and program section repeats are marked in a part program by labels.

A label is identified by a number between 1 and 254. Each label can be set only once with LABEL SET in a program.

If a label is set more than once, the TNC sends an error message at the end of the LBL SET block.

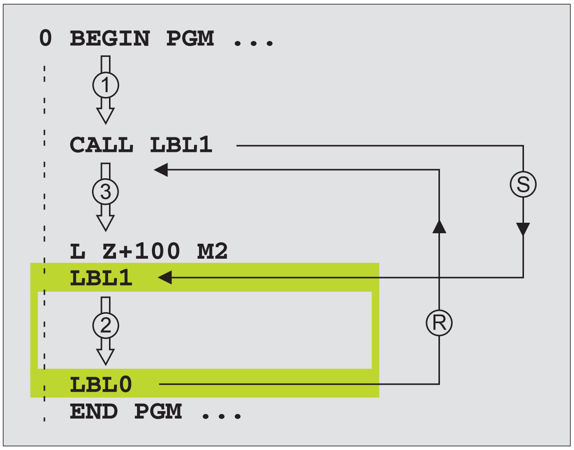
LABEL 0 (LBL 0) is used exclusively to mark the end of a subprogram and can therefore be used as often as desired.

For a better overview in this program window, LBL blocks and CALL LBL blocks are indented to the left by one character.

9.2 Subprograms

Operating sequence

1. The TNC executes the part program up to the block in which a subprogram is called with CALL LBL.
2. The subprogram is then executed from beginning to end. The subprogram end is marked LBL 0.
3. The TNC then resumes the part program from the block after the subprogram call.



Programming notes

- A main program can contain up to 254 subprograms.
- You can call subprograms in any sequence and as often as desired.
- A subprogram cannot call itself.
- Write subprograms at the end of the main program (behind the block with M2 or M30).
- If subprograms are located before the block with M02 or M30, they will be executed at least once even if they are not called.

Programming a subprogram

LBL SET

- To mark the beginning, press the LBL SET key and enter a label number.
- Enter the subprogram number.
- To mark the end, press the LBL SET key and enter the label number "0".

Calling a subprogram

LBL CALL

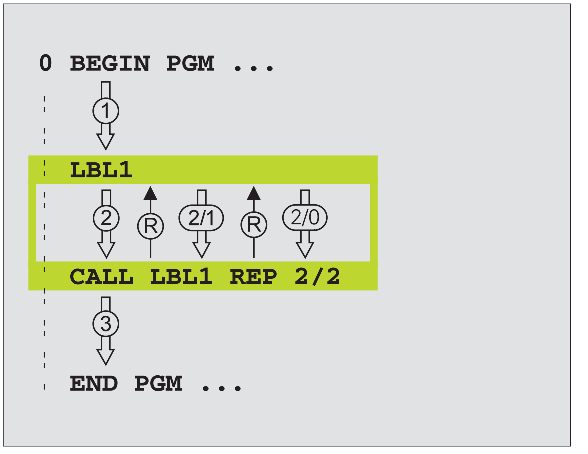
- To call a subprogram, press the LBL CALL key.
- **`Label number:`** Enter the label number of the subprogram you wish to call.
- **`Repeat REP:`** Ignore the dialog question with the NO ENT key. Repeat REP is used only for program section repeats.

CALL LBL 0 is not permitted (label 0 is only used to mark the end of a subprogram).

9.3 Program Section Repeats

Label LBL

The beginning of a program section repeat is marked by the label LBL. The end of a program section repeat is identified by CALL LBL /REP.



Operating sequence

1 The TNC executes the part program up to the end of the program section (CALL LBL /REP).
2 Then the program section between the called LBL and the label call is repeated the number of times entered after REP.
3 The TNC then resumes the part program after the last repetition.

Programming notes

- You can repeat a program section up to 65 534 times in succession.
- The number behind the slash after REP indicates the number of repetitions remaining to be run.
- The total number of times the program section is executed is always one more than the programmed number of repeats.

Resetting the program repeat counters after an interruption

If you interrupt program run during a program section repeat and then restart, the TNC resets the program section repeat counters as follows:

- If you select a new program, the TNC resets all counters.
- If you restart the same program with GOTO 0, the TNC resets the counters in the current program.
- If you do not return to the start of the program (GOTO > 0), the TNC does not reset any counters.

Programming a program section repeat

LBL SET

- To mark the beginning, press the LBL SET key and enter a LABEL NUMBER for the program section you wish to repeat.
- Enter the program section.

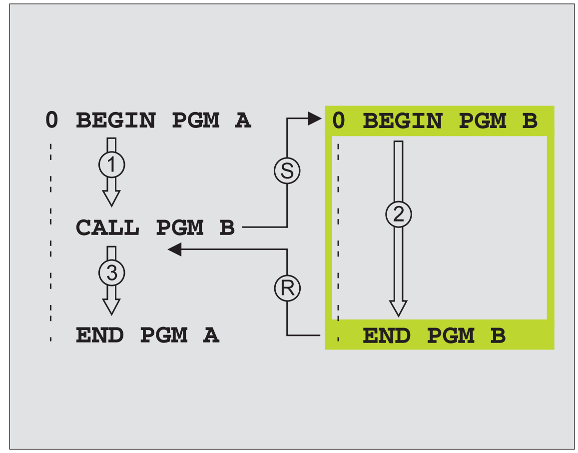
Calling a program section repeat

LBL CALL

▶ Press the LBL CALL key and enter the label number of the program section you want to repeat as well as the number of repeats (with Repeat REP).

9.4 Separate Program as Subprogram

Operating sequence



1 The TNC executes the part program up to the block in which another program is called with CALL PGM.

2 Then the other program is run from beginning to end.

3 The TNC then resumes the first (calling) part program with the block behind the program call.

Programming notes

- Programs called from external storage media must not contain subprograms or program section repeats.
- No labels are needed to call any program as a subprogram.
- The called program must not contain the miscellaneous functions M2 or M30.
- The called program must not contain a program call into the calling program, otherwise an infinite loop will result.

Calling any program as a subprogram

▶ To select the functions for program call, press the PGM CALL key.

▶ **Program Name:** Enter the name of the program to be called.

You can also call a program with Cycle 12 PGM CALL (see also "Calling Cycle 12 PGM-CALL" on page 171).

9.5 Nesting

Types of nesting

- 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

Example NC blocks

0 BEGIN PGM SUBPGMS MM	
...	
17 CALL LBL 1	Calling a subprogram at LBL 1
...	
35 L Z+100 R0 FMAX M2	Last program block of the
	main program (with M2)
36 LBL 1	Beginning of subprogram 1
...	
39 CALL LBL 2	Call the subprogram marked with LBL2
...	
45 LBL 0	End of subprogram 1
46 LBL 2	Beginning of subprogram 2
...	
62 LBL 0	End of subprogram 2
63 END PGM SUBPGMS MM	

Program execution

1. Main program SUBPGMS is executed up to block 17.
2. Subprogram 1 is called, and executed up to block 39.
3. 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.
4. Subprogram 1 is executed from block 40 up to block 45. End of subprogram 1 and return jump to the main program SUBPGMS.
5. Main program SUBPGMS is executed from block 18 up to block 35. Return jump to block 1 and end of program.

A subprogram that ends with LBL 0 cannot be located within another subprogram.

Repeating program section repeats

Example NC blocks

0 BEGIN PGM REPS MM	
...	
15 LBL 1	Beginning of program section repeat 1
...	
20 LBL 2	Beginning of program section repeat 2
...	
27 CALL LBL 2 REP 2/2	The program section between this block and LBL 2
...	(block 20) is repeated twice
35 CALL LBL 1 REP 1/1	The program section between this block and LBL 1
...	(block 15) is repeated once.
50 END PGM REPS MM	

Program execution

1. Main program REPS is executed up to block 27.
2. Program section between block 27 and block 20 is repeated twice.
3. Main program REPS is executed from block 28 to block 35.
4. Program section between block 35 and block 15 is repeated once (including the program section repeat between 20 and block 27).
5. Main program REPS is executed from block 36 to block 50 (end of program).

Repeating a subprogram

Example NC blocks

`0 BEGIN PGM SUBREP MM`	
`...`	
`10 LBL 1`	Beginning of program section repeat 1
`11 CALL LBL 2`	Subprogram call
`12 CALL LBL 1 REP 2/2`	The program section between this block and LBL 1
`...`	(block 10) is repeated twice
`19 L Z+100 R0 FMAX M2`	Last program block of the main program with M2
`20 LBL 2`	Beginning of subprogram
`...`	
`28 LBL 0`	End of subprogram
`29 END PGM SUBREP MM`	

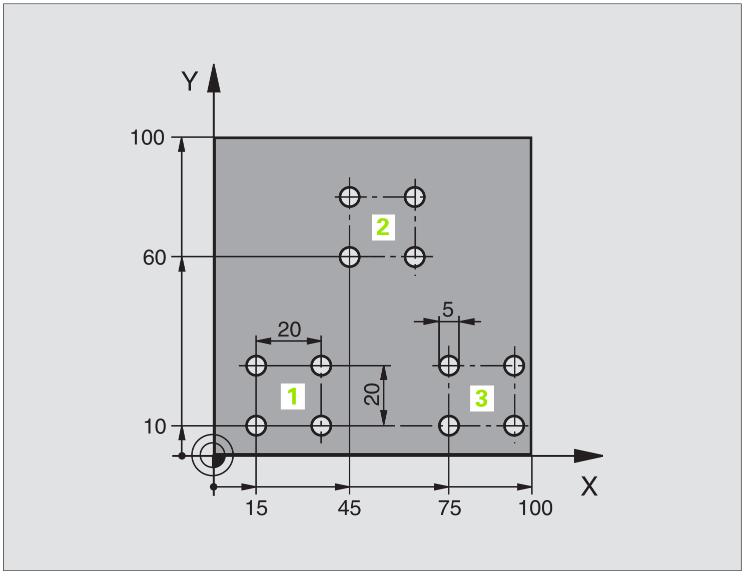
Program execution

1. Main program SUBREP is executed up to block 11.
2. Subprogram 2 is called and executed.
3. Program section between block 12 and block 10 is repeated twice. This means that subprogram 2 is repeated twice.
4. Main program SUBREP is executed once from block 13 to block 19. End of program.

Example: Erosion hole patterns

Program sequence

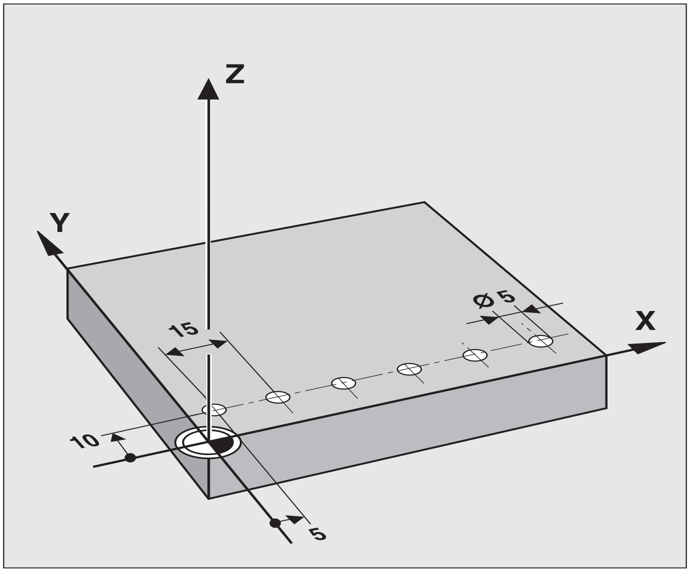
- Approach the erosion hole patterns in the main program
- Call the erosion hole pattern (subprogram 1)
- Program the erosion hole pattern only once in subprogram 1



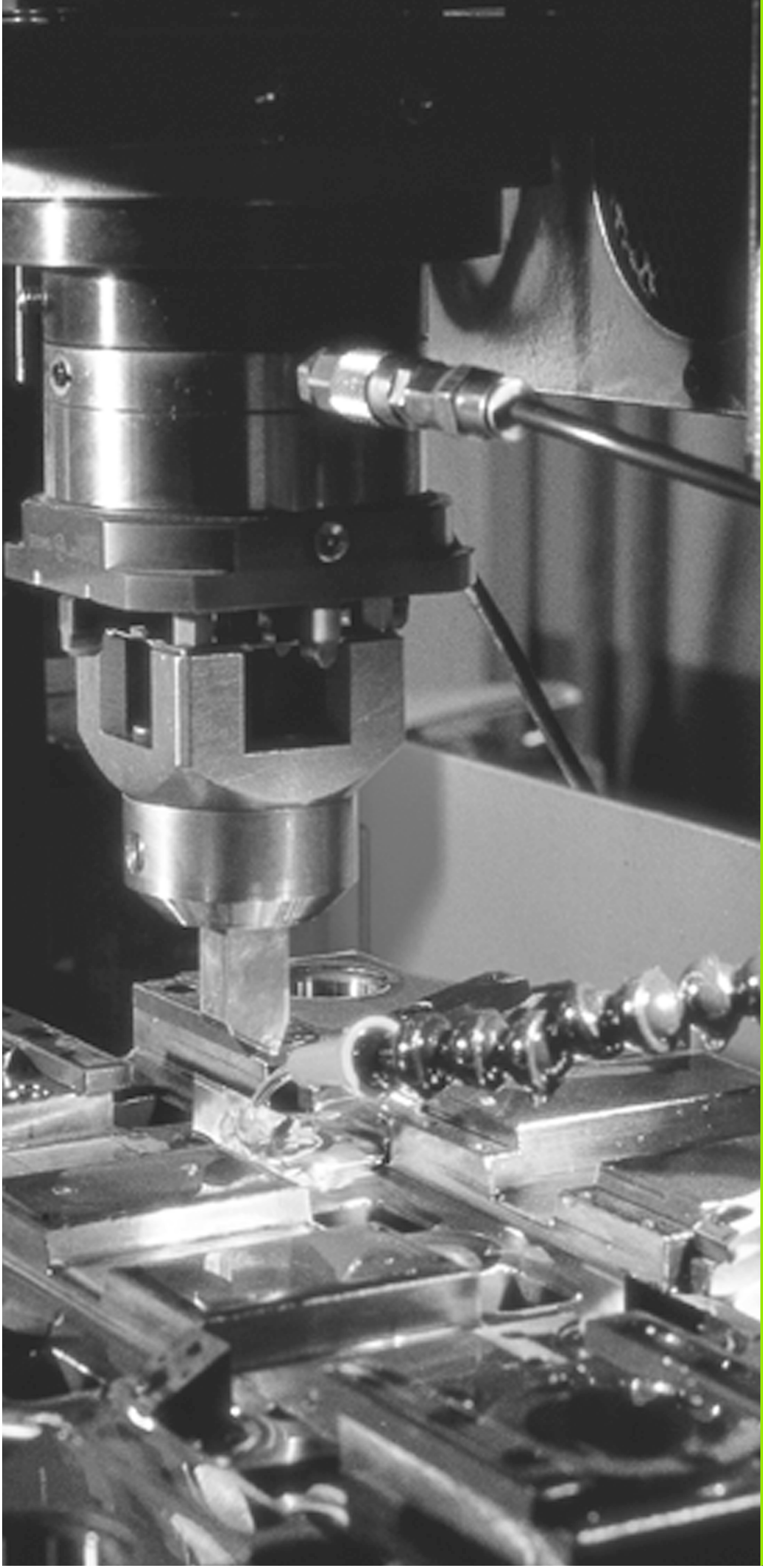
0 BEGIN PGM GROUPS MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Define the blank
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 CYCL DEF 1.0 GENERATOR	Cycle GENERATOR (see "Cycle 1 GENERATOR" on page 133)
4 CYCL DEF 1.1 P-TAB 10	Select erosion table (here, table 10)
5 CYCL DEF 1.2 MAX=10 MIN=10	Set power stage (here, to stage 10)
6 TOOL DEF 1 L+0 R+2.4	Define electrode in the program
7 TOOL CALL 1 Z U+0.2	Call electrode in the infeed axis Z, undersize 0.2 mm
8 L Z+100 R0 F MAX M	Retract in the infeed axis; rapid traverse; insert electrode
9 L X+15 Y+10 R0 F MAX M	Move to erosion hole group 1; rapid traverse
10 L Z+2 R FMAX M	Pre-position in the infeed axis
11 CALL LBL 1	Subprogram call (the subprogram is executed once with block 11)
12 L X+45 Y+60 R F MAX M	Move to erosion hole group 2
13 CALL LBL 1	Call subprogram 1
14 L X+75 Y+10 R0 F MAX M	Move to erosion hole group 3
15 CALL LBL 1	Call subprogram 1
16 L Z+100 R F MAX M2	Retract electrode; end of main program (M2);
	Subprograms are entered after M2
17 LBL 1	Beginning of subprogram 1
18 CALL LBL 2	Call subprogram 2

19 L IX+20 R F MAX M	Move to second cavity
20 CALL LBL 2	Call subprogram 2
21 L IY+20 R F MAX M	Move to third cavity
22 CALL LBL 2	Call subprogram 2
23 L IX–20 R F MAX M	Move to fourth cavity
24 CALL LBL 2	Call subprogram 2
25 LBL 0	End of subprogram 1
26 LBL 2	Beginning of subprogram 2
27 L Z–10 R F M36	Sink; eroding ON
28 L Z+2 F MAX M37	Retract electrode; eroding OFF
29 LBL 0	End of subprogram 2
30 END PGM GROUPS MM	

Example: Erosion hole row parallel to X axis



0 BEGIN PGM ROW MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	
2 BLK FORM 0.2 X+100 Y+100 Z+0	Define the blank
3 CYCL DEF 1.0 GENERATOR	Cycle GENERATOR (see "Cycle 1 GENERATOR" on page 133)
4 CYCL DEF 1.1 P-TAB 10	Select erosion table (here, table 10)
5 CYCL DEF 1.2 MAX=8 MIN=8	Set power stage (here, to stage 8)
6 TOOL DEF 1 L+0 R+2.4	Define electrode in the program
7 TOOL CALL 1 Z U+0.1	Call electrode in the infeed axis Z, undersize 0.1 mm
8 L Z+100 R0 F MAX	Retract in the infeed axis; rapid traverse; insert electrode
9 L X-10 Y+10 Z+2 F MAX M3	Pre-position to eroding distance in negative X direction
10 LBL 1	Start of the program block to be repeated
11 L IX+15 F MAX	Position above the cavity; rapid traverse
12 L Z-10 M36	Sink; eroding ON
13 L Z+2 F MAX M37	Retract electrode; rapid traverse; eroding OFF
14 CALL LBL 1 REP 5/5	Call LBL 1; program section between block 10 and
	block 14 is repeated five times (for 6 cavities!)
15 L Z+100 R0 F MAX	Retract electrode
16 END PGM ROW MM	

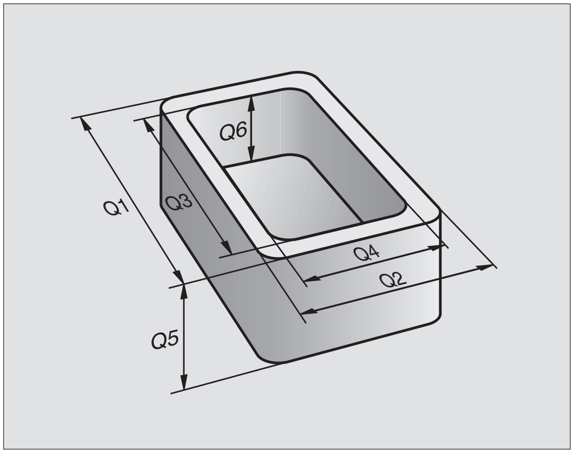


10

Programming: Q Parameters

10.1 Principle and Overview

You can program an entire family of parts in a single part program. You do this by entering variables called Q parameters instead of fixed numerical values.



Q parameters can represent information such as:

- Coordinate values
- Electrode data
- Cycle data

Q parameters are designated by the letter Q and a number between 0 and 255.

Q parameters also enable you to program **contours** that are defined through **mathematical functions.**In addition, you can use Q parameters to make execution of machining steps depend on certain **logical conditions.** You can mix
Q parameters and **fixed numerical values** within a program.

Some Q parameters are always assigned the same data by the TNC. For example, Q108 is always assigned the current electrode radius (see ”Preassigned Q parameters” on page 202).

Automatic deletion of Q parameters

The TNC deletes Q parameters (and the status display) when user parameter 7300 = 1 and the miscellaneous functions M00, M02 or M30 or the END block are executed in a program.

10.2 Part Families – Q Parameters in Place of Numerical Values

The Q parameter function FN0: ASSIGN assigns numerical values to Q parameters. This enables you to use variables in the program instead of fixed numerical values.

Example NC blocks

`15 FN0: Q10=25`	Assign
`...`	Q10 contains the value 25
`25 L X +Q10`	Means L X +25

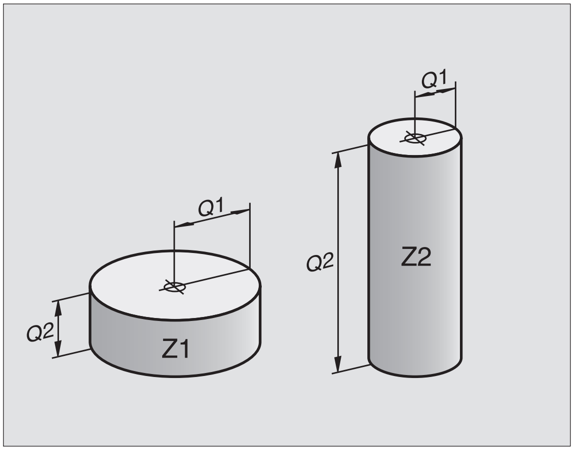
You need write only one program for a whole family of parts, entering the characteristic dimensions as Q parameters.

To program a particular part, you then assign the appropriate values to the individual Q parameters.

Example

Cylinder with Q parameters

Cylinder radius	R = Q1
Cylinder height	H = Q2
Cylinder Z1	Q1 = +30 Q2 = +10
Cylinder Z2	Q1 = +10 Q2 = +50



To assign numerical values to Q parameters

Example:

Q	To select Q parameter functions, press the Q key.

`FN0: ASSIGN`

ENT	Select FN 0: ASSIGN

`PARAMETER NUMBER FOR RESULT ?`

5 ENT	Enter the number of the Q parameter: 5

`FIRST VALUE OR PARAMETER ?`

10 ENT	Assign a value to Q5, for example 10.

Example NC block

```
FN0: Q5 = 10
```

The TNC assigns the numerical value on the right of the equal sign to the Q parameter on the left of the equal sign.

10.3 Describing Contours through Mathematical Operations

Function

The Q parameters listed below enable you to program basic mathematical functions in a part program:

- Select a Q parameter function: Press the Q key (in the numerical keypad at right). The dialog **FN 0: Assign** appears.
- Select a function directly: Press GOTO, enter the function number, and select it with the ENT key, or
- Select a function with the arrow keys: Use the arrow keys to select the desired function, and confirm your selection with the ENT key.

Overview

Function
FN0: ASSIGN Example: **FN0: Q5 = +60** Assigns a numerical value.
FN1: ADDITION Example: **FN1: Q1 = –Q2 + –5** Calculates and assigns the sum of two values.
FN2: SUBTRACTION Example: **FN2: Q1 = +10 – +5** Calculates and assigns the difference of two values.
FN3: MULTIPLICATION Example: **FN3: Q2 = +3 * +3** Calculates and assigns the product of two values.
FN4: DIVISION Example: **FN4: Q4 = +8 DIV +Q2** Calculates and assigns the quotient of two values. **Not permitted:** division by 0
FN5: SQUARE ROOT Example: **FN5: Q20 = SQRT 4** Calculates and assigns the square root of a number. **Not permitted:** Square root of a negative number

To the right of the "=" character you can enter the following:

- Two numbers
- Two Q parameters
- A number and a Q parameter

The Q parameters and numerical values in the equations can be entered with positive or negative signs.

Programming example for basic mathematical operations

Assign the value 10 to the parameter Q5:

Q — To select Q parameter functions, press the Q key.

FN 0: ASSIGN

ENT — To select the Q parameter function FN0, press the ENT key.

PARAMETER NUMBER FOR RESULT ?

5 ENT — Enter the number of the Q parameter: 5

FIRST VALUE OR PARAMETER ?

10 ENT — Assign the value 10 to Q5.

Example: Program blocks in the TNC

```
16 FN0: Q5 = +10
17 FN3: Q12 = +Q5 * +7
```

Assign the product of Q5 and Q7 to Q12:

Q	To select Q parameter functions, press the Q key.

FN 0: ASSIGN

GOTO 3	Select the function directly: Press GOTO and enter the number of the function, for example FN3, or
↑ ↓	Select the function with the arrow keys.

FN 3: MULTIPLICATION

ENT	Select FN 3: MULTIPLICATION

PARAMETER NUMBER FOR RESULT ?

12 ENT	Enter the number of the Q parameter: 12

FIRST VALUE OR PARAMETER ?

Q5 ENT	Enter Q5 for the first value.

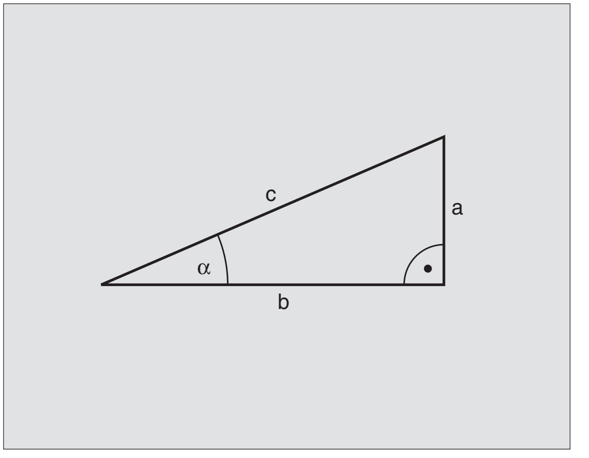
SECOND VALUE OR PARAMETER ?

Q7 ENT	Enter 7 for the second value.

10.4 Trigonometric Functions

Definitions

Sine, cosine and tangent are terms designating the ratios of sides of right triangles. For a right triangle, the trigonometric functions of the angle a are defined by the following equations:



Sine: $\sin \alpha = a / c$
Cosine: $\cos \alpha = b / c$
Tangent: $\tan \alpha = a / b = \sin \alpha / \cos \alpha$

where

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

The TNC can find the angle from the tangent:

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

Example:

a = 10 mm

b = 10 mm

$\alpha = \arctan (a / b) = \arctan 1 = 45°$

Furthermore:

$a^2 + b^2 = c^2$ (where $a^2 = a \times a$)

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

Overview of functions

Function
FN6: SINE Example: **`FN6: Q20 = SIN–Q5`** Calculate the sine of an angle in degrees (°) and assign it to a parameter.
FN7: COSINE Example: **`FN7: Q21 = COS–Q5`** Calculate the cosine of an angle in degrees (°) and assign it to a parameter.
FN8: ROOT SUM OF SQUARES Example: **`FN8: Q10 = +5 LEN +4`** Calculate and assign length from two values.
FN13: ANGLE Example: **`FN13: Q20 = +10 ANG–Q1`** Calculate the angle from the arc tangent of two sides or from the sine and cosine of the angle (0 < angle < 360°) and assign it to a parameter.

10.5 If-Then Decisions with Q Parameters

Function

The TNC can make logical If-Then decisions by comparing a Q parameter with another Q parameter or with a numerical value. If the condition is fulfilled, the TNC continues the program at the label that is programmed after the condition (for information on labels see also "Labeling Subprograms and Program Section Repeats" on page 174). If it is not fulfilled, the TNC continues with the next block.

To call another program as a subprogram, enter PGM CALL after the block with the target label.

Unconditional jumps

An unconditional jump is programmed by entering a conditional jump whose condition is always true. Example:

FN9: IF+10 EQU+10 GOTO LBL1

Programming If-Then decisions

If-Then decisions appear when the Q function key is pressed, and after selection with the GOTO key or the arrow keys. The TNC displays the following dialogs:

Function
FN9: IF EQUAL, JUMP Example: **`FN9: IF +Q1 EQU +Q3 GOTO LBL 5`** If the two values or parameters are equal, jump to the given label.
FN10: IF NOT EQUAL, JUMP Example: **`FN10: IF +10 NE –Q5 GOTO LBL 10`** If the two values or parameters are not equal, jump to the given label.
FN11: IF GREATER THAN, JUMP Example: **`FN11: IF+Q1 GT+10 GOTO LBL 5`** If the first parameter or value is greater than the second value or parameter, jump to the given label.
FN12: IF LESS THAN, JUMP Example: **`FN12: IF+Q5 LT+0 GOTO LBL 1`** If the first value or parameter is less than the second value or parameter, jump to the given label.

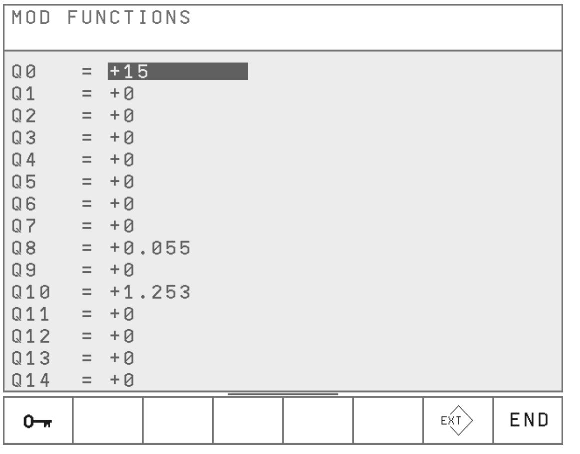
Abbreviations used:

IF	:	If
EQU	:	Equals
NE	:	Not equal
GT	:	Greater than
LT	:	Less than
GOTO	:	Go to

10.6 Checking and Changing Q Parameters

Procedure

Q parameters can be checked during a program run or test run. If you wish to change any Q parameters, you must interrupt the program run or test run.



Select the supplementary operating mode MOD.

Q-PAR

Press the Q parameter status soft-key: The TNC shows a list of the first 15 parameters.

To scroll through the subsequent Q parameters, press and hold the down arrow key. You can go to a specific Q parameter by pressing the GOTO key and entering the desired Q parameter number.

0 ENT

You can only change the Q parameter if you have interrupted the program run or test run. Enter the new value, for example 0, and confirm with the ENT key.

Return to the last active operating mode.

10.7 Output of Q Parameters and Messages

Output of error messages

With the function FN14: ERROR you can call messages that were preprogrammed by the machine tool builder.

When the TNC encounters a block with FN 14 during program run, it interrupts the run and displays an error message. The program must then be restarted.

Input:

Example: FN 14: ERROR = 254

The TNC then displays the text stored under error number 254.

Range of error numbers	Standard dialog text
0 ... 299	FN 14: Error code 0 299
300 ... 799	PLC dialogs from 0....499

Example NC block

```
180 FN14: ERROR = 254
```

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

Output through an external data interface

The function FN15: PRINT transfers Q parameter values and error messages through the data interface, for example to a printer or to the file %FN15RUN.A.

- FN 15: PRINT with numerical values from 0 to 499 are used to access PLC dialogs 0 to 499.
  Example: FN 15: PRINT 20
  Transfers the error message (see overview at FN 14).
- FN 15: PRINT with numerical value 200
  Example: FN 15: PRINT 200
  Transfers the ETX character (end of text).
- FN 15: PRINT with Q parameters Q1 to Q255
  Example: FN 15: PRINT Q20
  Transfers the value of the Q parameter.

You can transfer up to six Q parameters and numerical values simultaneously. The TNC separates them with slashes.

Example NC block

```
23 FN 15: PRINT  1/Q1/2/Q2
```

Indexed assignment

The function FN16: INDEXED DATA ASSIGNMENT accesses a Q parameter in a previously created list (for example, a list of gap diameter values).
In the following example, Q55 is the **pointer parameter** that points to a Q parameter in a list, and Q200 is the **base parameter** that indicates the beginning of the list.

Example:

```
12 FN 0: Q55 = 5
...
23 FN 16: Q20 = Q200 (Q55)
...
```

```
Q200<-- Q20
Q201 = 0.04
Q202 = 0.08
Q203 = 0.12
Q204 = 0.16
Q205 = 0.20<-- Q55
Q206 = 0.24
Q207 = 0.28
.....
```

The TNC assigns to Q parameter Q20 the value that is in the fifth position in the list from Q200.

Transferring values to/from the PLC

With the function FN 19: PLC you can send data to the PLC or receive data from the PLC.

Example:

```
22 FN 19: PLC + 11/+Q13/Q77
```

The value 11 is transferred to word D280.
The contents of Q parameter Q13 are transferred to word D284 (optional entry, can be ignored with NO ENT).
The value from word D512 is transferred to Q parameter Q77 by the PLC and can now be evaluated in the subsequent NC part.

10.8 Measuring with a probing electrode during program run

Introduction

You can use a probing electrode to probe positions on the workpiece during program run.

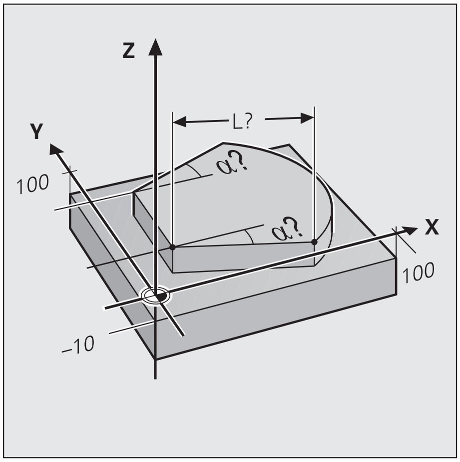
Applications:

- Measuring differences in the height of cast surfaces
- Tolerance checking during machining

To program the use of a probing electrode, press the TOUCH PROBE key.

You pre-position the electrode to automatically probe the desired position. The coordinate measured for the probe point is stored in a Q parameter.
The TNC interrupts the probing process if the electrode does not reach the workpiece within a certain distance (programmed in MP6130).

The C axis can also be defined as the electrode axis.



To program the use of a probing electrode

TOUCH PROBE — Select the probing function: Press the TOUCH PROBE key.

TCH PROBE 0: REF. PLANE

ENT — To select the touch probe functions:

PARAMETER NUMBER FOR RESULT?

5 ENT — Enter the number of the Q parameter that you want to assign the coordinate to, e.g. Q5.

PROBING AXIS / PROBING DIRECTION?

X — Enter the probing axis for the coordinate, e.g. X.

−/+ ENT — Select the probing direction and confirm it.

POSITION VALUE?

X 5 — Enter all coordinates of the pre-positioning point of the electrode, e.g. X = 5 mm, Y = 1 mm, Z = –5 mm.

Y 1

−/+ 5

ENT — Concludes your input.

Example NC blocks

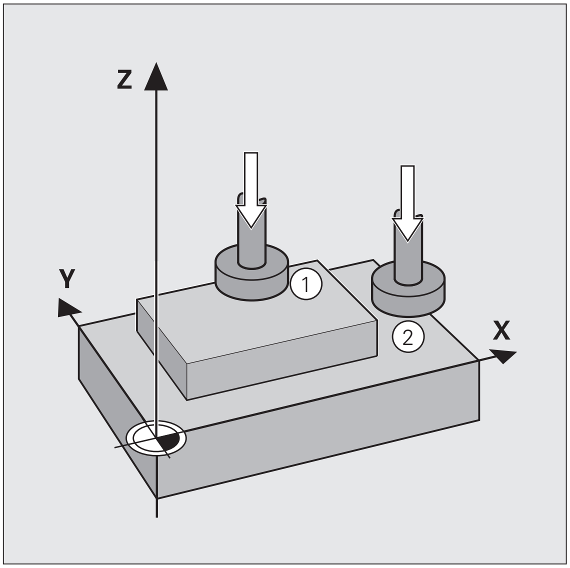
```
180 TCH PROBE 0.0 REF. PLANE Q5 X-
181 TCH PROBE 0.1 X+5 Y+0 Z-5
```

- Pre-position the electrode manually to avoid a collision when the programmed pre-positioning point is approached.
- You can also use the programmable probing function when the "Tilt working plane" function is active. The TNC then acknowledges the coordinate of the touch point in the tilted coordinate system.

Example: Measuring the height of an island on a workpiece

Program sequence

- Store coordinates for pre-positioning the electrode in Q parameters
- Probe probe point 1
- Probe probe point 2
- Determine the height from the difference in Z values



0 BEGIN PGM PROBE MM	
1 FN 0: Q11 = +20	Parameter coordinates for probe point 1 in the X axis
2 FN 0: Q12 = +50	Parameter coordinates for probe point 1 in the Y axis
3 FN 0: Q13 = +10	Parameter coordinates for probe point 1 in the Z axis
4 FN 0: Q21 = +50	Parameter coordinates for probe point 2 in the X axis
5 FN 0: Q22 = +10	Parameter coordinates for probe point 2 in the Y axis
6 FN 0: Q23 = +0	Parameter coordinates for probe point 2 in the Z axis
7 TOOL CALL 0 Z	Insert probing electrode
8 L Z+100 R0 F MAX M	Retract to safety clearance
9 TCH PROBE 0.0 REF. PLANE Q10 Z–	Assign the Z coordinate probed in the negative direction to Q10
10 TCH PROBE 0.1 X+Q11 Y+Q12 Z+Q13	Touch probe is valid for point 1
11 L X+Q21 Y+Q22	Auxiliary point for second pre-positioning
12 TCH PROBE 0.0 REF. PLANE Q20 Z–	Assign the Z coordinate probed in the negative direction to Q20
13 TCH PROBE 0.1 X+Q21 Y+Q22 Z+Q23	Touch probe is valid for point 2
14 FN2: Q1 = Q10–Q20	Measure the height of the island and assign to Q1
15 STOP	Q1 can be checked after program run has stopped
16 L Z+100 R0 F MAX M2	Retract probing electrode and end program
17 END PGM PROBE MM	End of program

10.9 Q Parameters with Special Functions

Vacant Q parameters

Q parameters Q0 to Q79 are freely programmable.
The TNC always uses the last numerical value assigned to these Q parameters (see Chapter 8).

When programming part families with Q parameters, you should only use "vacant" Q parameters. This ensures that the TNC does not overwrite a parameter used in the program.

Preassigned Q parameters

The TNC always assigns the same values to the following Q parameters, e.g. the electrode radius or the current generator power stage.

Q parameters with special functions

Some Q parameters have special functions. For example, the TNC uses these parameters to transfer values between the program and the datum table:
Q80 to Q84

Preassigned Q parameters

Additional erosion parameters: Q96, Q97, Q98

If you work with erosion tables, the machine tool builder can store additional erosion parameters in the Q parameters Q96, Q97 and Q98.

The machine tool builder can give you more information about these Q parameters.

Data from the erosion table

When you are working with an erosion table, the following erosion parameters are also available in Q parameters.

Erosion parameters	Parameters
Current power stage LS	Q99
Surface finish [µm]	Q148
Highest power stage	Q150
Lowest power stage	Q151
Number of the active erosion table	Q152
Minimum undersize UNS of the lowest power stage [mm]	Q154
Two-times gap 2G of the lowest power stage [mm]	Q155
Two-times gap 2G of the highest power stage [mm]	Q156
Two-times gap 2G from the lowest to the highest power stage [mm]	Q201 to Q225
Minimum undersize UNS of the lowest to the highest power stage [mm]	Q231 to Q255

Q parameters when not using erosion tables: Q90 to Q99

If you are working without erosion tables, you must use the Q parameters for eroding (Q90 to Q99).

The machine tool builder can give you more information about these Q parameters.

Electrode data: Q108, Q158 to Q160

The TNC stores the electrode data that you entered in the TOOL DEF, TOOL CALL and EL-CORR blocks in the following Q parameters:

Entry	Parameters
Electrode radius from TOOL DEF	Q108
Electrode undersize from TOOL CALL	Q158
Electrode length from TOOL DEF	Q159
Electrode number from TOOL CALL	Q160

Electrode axis Q109

The value of parameter 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

Miscellaneous functions for free rotation of the C axis: Q110

The value of parameter Q110 depends on which M function was last programmed for the rotation of the C axis:

Definition of miscellaneous functions	Parameter value
No M3, M4 or M5 defined	Q110 = –1
M03: Free rotation of C axis ON	Q110 = 0
M04: Free rotation of C axis OFF	Q110 = 1
M05 active	Q110 = 2

Flushing: Q111

M functions	Parameter value
Directly after program selection	Q111 = –1
Flushing OFF (M09 active)	Q111 = 0
Flushing ON (M08 active)	Q111 = 1

Plane of rotation during ROTATION Cycle: Q112

Plane of rotation	Parameter value
No plane defined	Q112 = –1
Y/Z plane	Q112 = 0
Z/X plane	Q112 = 1
X/Y plane	Q112 = 2

Dimensions of the main program: Q113

Dimensions of the main program	Parameter value
Directly after program selection	Q113 = –1
Metric system (mm)	Q113 = 0
Inch system (inches)	Q113 = 1

Dimensions in the erosion table: Q114

Dimensions in the erosion table	Parameter value
Directly after table selection	Q114 = –1
Metric system (mm)	Q114 = 0
Inch system (inches)	Q114 = 1

Coordinates after probing during program run: Q115 to Q119

The parameters Q115 to Q119 contain the coordinates of the spindle position in the **machine system** at the moment of contact during programmed measurement with the probing electrode.
The length and radius of the probing electrode are ignored in these coordinates:

Coordinate axis	Parameter
X axis	Q115
Y axis	Q116
Z axis	Q117
IVth axis	Q118
Vth axis	Q119

Coordinates after probing during program run: Q120 to Q124

The parameters Q120 to Q124 contain the coordinates of the spindle position in the **workpiece system** at the moment of contact during programmed measurement with the probing electrode.
The length and radius of the probing electrode are ignored in these coordinates:

Coordinate axis	Parameter
X axis	Q120
Y axis	Q121
Z axis	Q122
IVth axis	Q123
Vth axis	Q124

Status for eroding with time limit: Q153

The TNC assigns values to the Q parameter Q153 if you are machining with Cycle 2 ERO.TIME LIM.:

Data	Parameter value
Return jump to the main program, for example, from the subprogram.	Q153 = 0
Time exceeded during eroding and Cycle 17 DISK cancelled.	Q153 = 1
Cycle 2 ERO.TIME LIM. completed.	Q153 = 2

Data about following electrode: Q157

Entry	Parameter value
Following electrode = YES	Q157 = 1
Following electrode = NO	Q157 = MP2040

Number of the cycle called with CYCL CALL: Q162

Entry	Parameter
Cycle number	Q162

Gap size LS max when machining which Cycle 1 GENERATOR: Q164

Entry	Parameter
Gap size	Q164

Q parameters with special functions

The TNC uses some Q parameters, for example, to exchange coordinates between the datum table or the integrated PLC and the program.

Q parameters for the datum table: Q81 to Q84

The TNC exchanges coordinates between the datum table and the machining program with the following Q parameters:

Datum coordinates	Parameter
Number of the datum in the table	Q80
X coordinate	Q81
Y coordinate	Q82
Z coordinate	Q83
C coordinate	Q84
Coordinate of the fifth axis	Q85

Q parameters from the PLC: Q100 to Q107

The TNC can assume preassigned Q parameters from the integrated PLC (Q100 to Q107).
The machine tool builder can give you more information about these Q parameters.

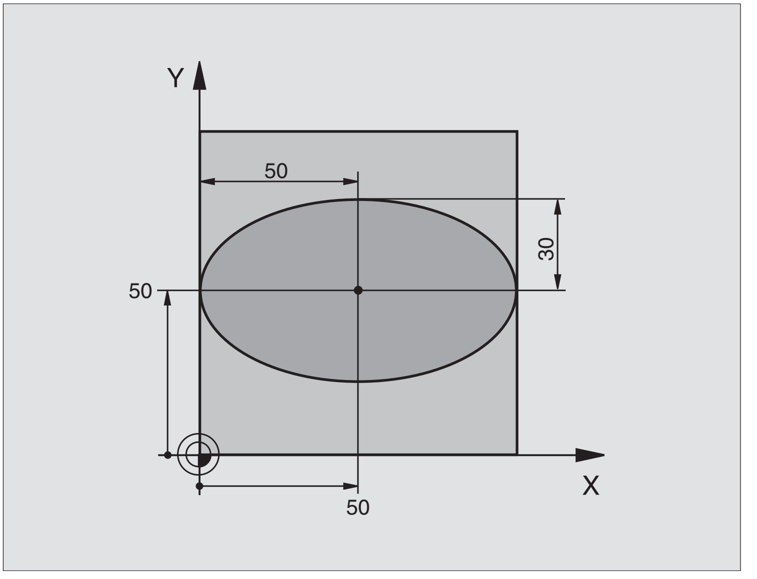
Machining time: Q161

The TNC stores the current machining time in Q parameter Q161.
Format: hh:mm:ss

Example: Ellipse

Program sequence

- The contour of the ellipse is approximated by many short lines (defined in Q7). The more calculating steps you define for the lines, the smoother the curve becomes.
- The machining direction can be altered by changing the entries for the starting and end angles in the plane:
  Clockwise machining direction:
  starting angle > end angle
  Counterclockwise machining direction: starting angle < end angle
- The tool radius is not taken into account.



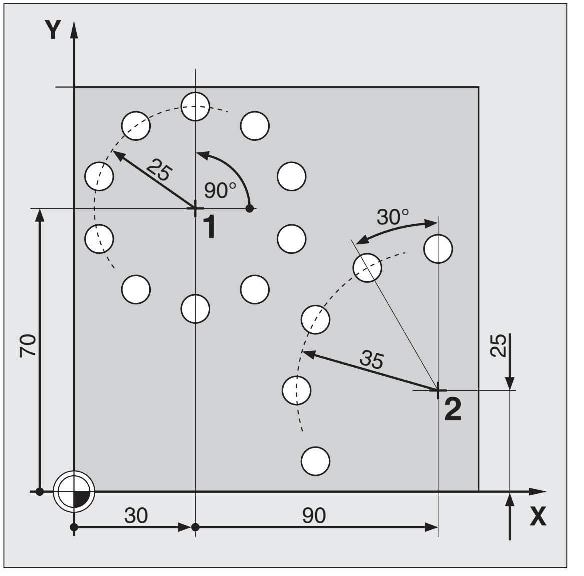
0 BEGIN PGM ELLIPSE MM	
1 FN 0: Q1 = +50	Center in X axis
2 FN 0: Q2 = +50	Center in Y axis
3 FN 0: Q3 = +50	Semiaxis in X
4 FN 0: Q4 = +30	Semiaxis in Y
5 FN 0: Q5 = +0	Starting angle in the plane
6 FN 0: Q6 = +360	End angle in the plane
7 FN 0: Q7 = +40	Number of calculating steps
8 FN 0: Q8 = +0	Rotational position of the ellipse
9 FN 0: Q9 = +5	Milling depth
10 FN 0: Q10 = +100	Feed rate for plunging
11 FN 0: Q11 = +350	Feed rate for milling
12 FN 0: Q12 = +2	Setup clearance for pre-positioning
13 BLK FORM 0.1 Z X+0 Y+0 Z-20	Define the workpiece blank
14 BLK FORM 0.2 X+100 Y+100 Z+0	
15 CYCL DEF 1.0 GENERATOR	Cycle GENERATOR (see "Cycle 1 GENERATOR" on page 133)
16 CYCL DEF 1.1 P-TAB 300	Select erosion table (here, table 300)
17 CYCL DEF 1.2 MAX=12 MIN=8	Power stages, for example between 8 and 12
18 TOOL DEF 1 L+0 R+5	Define electrode in the program
19 TOOL CALL 1 U+1	Call electrode in the infeed axis Z, undersize 1 mm

20 L Z+250 R0 F MAX M	Retract electrode
21 CALL LBL 10	Call machining operation
22 L Z+100 R0 F MAX M2	Retract in the tool axis, end program
23 LBL 10	Subprogram 10: Machining operation
24 CYCL DEF 7.0 DATUM SHIFT	Shift datum to center of ellipse
25 CYCL DEF 7.1 X+Q1	
26 CYCL DEF 7.2 Y+Q2	
27 CYCL DEF 10.0 ROTATION	Account for rotational position in the plane
28 CYCL DEF 10.1 ROT+Q8	
29 FN2 Q35 = Q6 - Q5	Starting angle - end angle
30 FN4 Q35 = Q35 DIV Q7	Calculate angle increment
31 FN0 Q37 = 0	Set counter
32 FN7 Q36 = COS Q5	
33 FN3 Q21 = Q3 * Q36	Calculate X coordinate for starting point
34 FN7 Q36 = SIN Q5	
35 FN3 Q22 = Q4 * Q36	Calculate Y coordinate for starting point
36 L X+Q21 Y+Q22 R0 F MAX M36	Move to starting point in the plane; eroding ON
37 L Z+Q12 R0 F MAX M	Pre-position in tool axis to setup clearance
38 L Z-Q9 R0 FQ10 M	Move to working depth
39 LBL 1	
40 FN1 Q36 = Q5 + Q35	Update the angle
41 FN1 Q37 = Q37 + 1	Update the counter
42 FN7 Q38 = COS Q36	
43 FN3 Q21 = Q3 * Q38	Calculate the current X coordinate
44 FN6 Q38 = SIN Q36	
45 FN3 Q22 = Q4 * Q38	Calculate the current Y coordinate
46 L X+Q21 Y+Q22 R0 FQ11 M	Move to next point
47 FN 12: IF +Q37 LT +Q7 GOTO LBL 1	Unfinished? If not finished, return to LBL 1
48 CYCL DEF 10.0 ROTATION	Reset the rotation
49 CYCL DEF 10.1 ROT+0	
50 CYCL DEF 7.0 DATUM SHIFT	Reset the datum shift
51 CYCL DEF 7.1 X+0	
52 CYCL DEF 7.2 Y+0	
53 L Z+Q12 R0 F MAX M37	Move to safety clearance; eroding OFF
54 LBL 0	End of subprogram
55 END PGM ELLIPSE MM	

Example: Circular hole patterns

Program sequence

- Define parameter coordinates for the full circle
- Define parameter coordinates for the circle arc
- The positions to be eroded are each approached in the subprogram LBL1 through movements in the plane with polar coordinates.



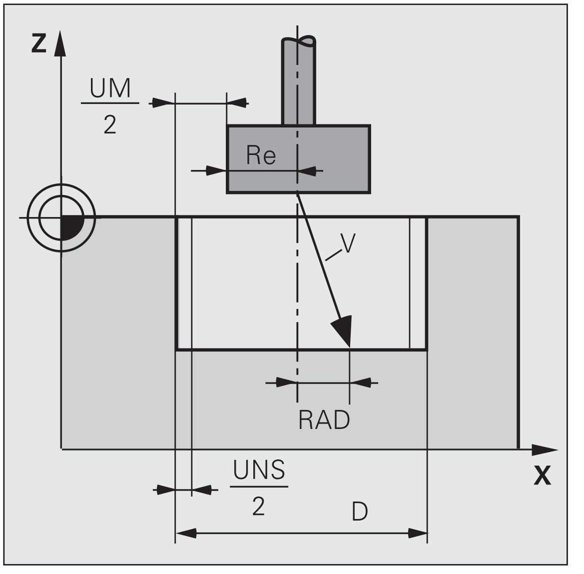
0 BEGIN PGM HOLES MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Define the workpiece blank: MIN point
2 BLK FORM 0.2 X+100 Y+100 Z+0	Define the workpiece blank: MAX point
3 CYCL DEF 1.0 GENERATOR	Cycle GENERATOR (see "Cycle 1 GENERATOR" on page 133)
4 CYCL DEF 1.1 P-TAB 300	Select erosion table (here, table 300)
5 CYCL DEF 1.2 MAX=12 MIN =12	Set power stage (here, to stage 12)
6 TOOL DEF 1 L+0 R+5	Define electrode in the program
7 TOOL CALL 1 Z U+1	Call electrode in the infeed axis Z, undersize 1 mm
8 L Z+100 R0 F MAX M	Retract in the infeed axis; rapid traverse; insert electrode
9 FN 0: Q1 = +30	Full circle 1: center X
10 FN 0: Q2 = +70	Full circle 1: center Y
11 FN 0: Q3 = +10	Full circle 1: number of cavities
12 FN 0: Q4 = +25	Full circle 1: radius
13 FN 0: Q5 = +90	1 and 2: starting angle
14 FN 0: Q6 = +0	Full circle 1: angle increment (input value 0: full circle)
15 FN 0: Q7 = +2	1 and 2: set-up clearance
16 FN 0: Q8 = -15	1 and 2: eroding depth
17 CALL LBL 1	Call subprogram 1 for full circle

18 FN 0: Q1 = +90	Circle segment 2: center X
19 FN 0: Q2 = +25	Circle segment 2: center Y
20 FN 0: Q3 = +5	Circle segment 2: number of cavities
21 FN 0: Q4 = +35	Circle segment 2: radius
22 FN 0: Q6 = +30	Circle segment 2: angle increment
23 CALL LBL 1	Call subprogram 1 for arc
24 LBL 1	Subprogram 1
25 FN 0: Q10 = +0	Reset counter for completed cavities
26 FN 10: IF +Q6 NE +0 GOTO LBL 10	If angle increment does not equal 0, go to LBL 10
27 FN 4: Q6 = +360 DIV +Q3	Calculate angle increment for full circle
28 LBL 10	
29 FN 1: Q11 = +Q5 + +Q6	Angle for second eroding position
30 CC X+Q1 Y+Q2	Position pole at center
31 LP PR+Q4 PA+Q5 R0 F MAX M3	Account for rotational position in the plane
32 L Z+Q7 R0 F MAX M	Pre-position electrode at set-up clearance
33 L Z+Q8 R F M36	First cavity; eroding ON
34 L Z+Q7 R F MAX M37	Retract electrode; eroding OFF
35 FN 1: Q10 = +Q10 + +1	Increment counter for completed cavities
36 FN 9: IF +Q10 EQU +Q3 GOTO LBL 99	If finished, jump to LBL 99
37 LBL 2	
38 LP PR+Q4 PA+Q11 R0 F MAX M	Move to next cavity
39 L Z+Q8 R F M36	Eroding
40 L Z+Q7 R F MAX M37	Retract electrode; eroding OFF
41 FN 1: Q10 = +Q10 + +1	Increment counter for completed cavities
42 FN 1: Q11 = +Q11 + +Q6	Calculate angle for next cavity
43 FN 12: IF +Q10 LT +Q3 GOTO LBL 2	If not yet finished, jump to LBL 2
44 LBL 99	
45 L Z+200 R F MAX M	Retract electrode
46 LBL 0	End of subprogram
47 END PGM HOLES MM	

Example: Cavity with DISK cycle

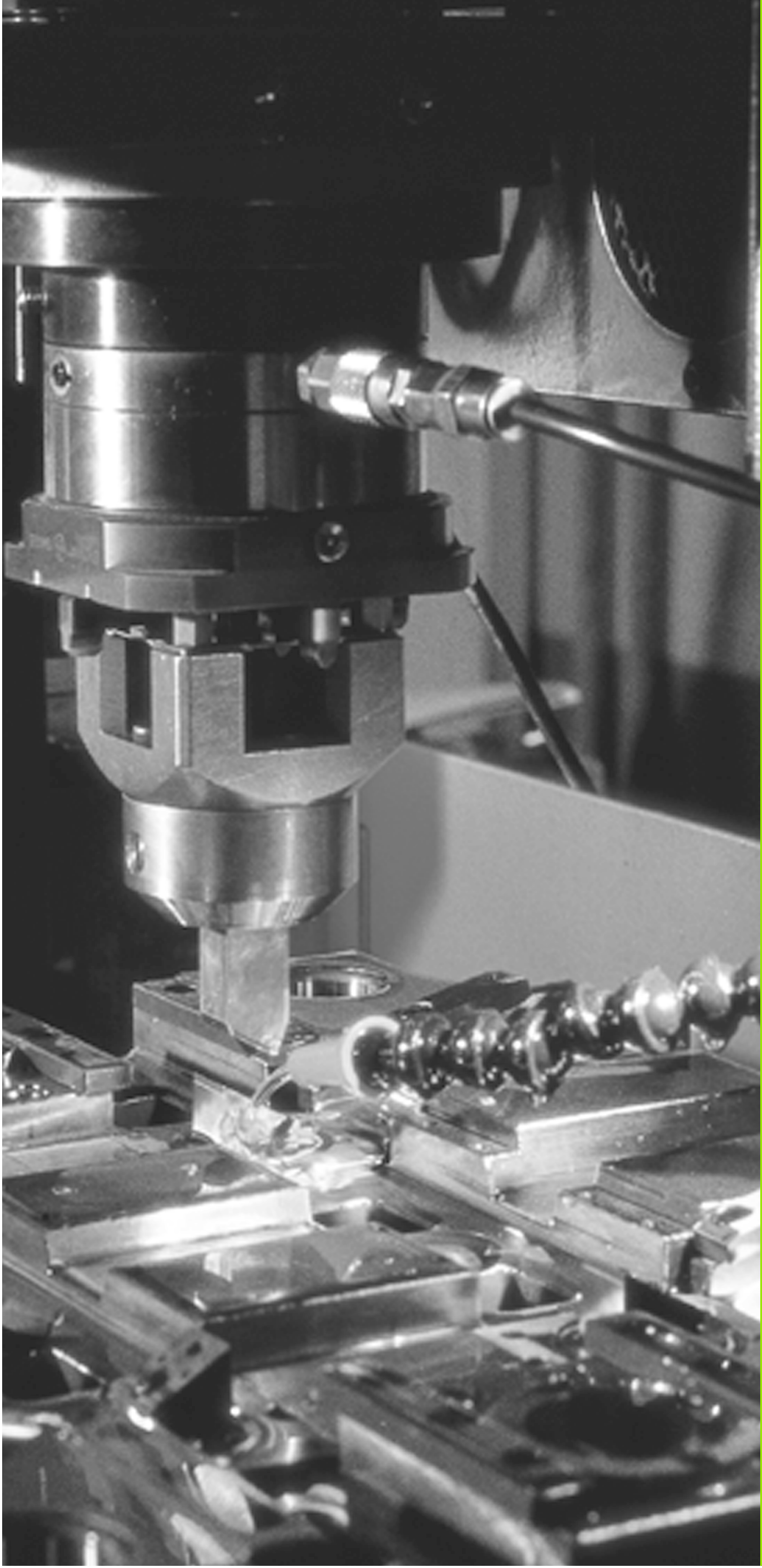
Program sequence

- The program accesses the minimum undersize UNS with indexed data assignment via the power stage.
- The minimum undersize UNS is located in the erosion table.
- Calculations in the program:
  Undersize UM = D – 2 • R
  Expansion radius RAD = 0.5 • (UM – UNS)
- The depth of cavity T is reduced by the value of UNS programmed in the DISK cycle. The electrode radius must be larger than the radius of the cavity (= 0.5 • D=20 mm).



0 BEGIN PGM QDISK MM	
1 FN 0: Q1 = – 10	Depth of cavity T
2 FN 0: Q2 = + 40	Diameter of cavity D
3 FN 0: Q99 = 6	Power stage
4 BLK FORM 0.1 Z X+0 Y+0 Z–20	Define the workpiece blank: MIN point
5 BLK FORM 0.2 X+100 Y+100 Z+0	Define the workpiece blank: MAX point
6 CYCL DEF 1.0 GENERATOR	Cycle GENERATOR (see “Cycle 1 GENERATOR” on page 133)
7 CYCL DEF 1.1 P-TAB 100	Select erosion table (here, table 100)
8 CYCL DEF 1.2 MAX=6 MIN =6	Set power stage
9 TOOL DEF 1 L+0 R+12	Define electrode in the program
10 TOOL CALL 1 Z U+1	Call electrode in the infeed axis Z, undersize 1 mm
11 L Z+100 R0 F MAX M	Retract in the infeed axis; rapid traverse; insert electrode
12 L X+50 Y+50 Z+2 R0 FMAX M	Starting position
13 FN 3: Q10 = +2 * +Q108	Calculate electrode diameter
14 FN 2: Q10 = +Q2 – +Q10	Calculate undersize UM
15 TOOL CALL 1 Z UM +Q10	Call tool with UM
16 FN 16: Q11 = Q231(Q99)	Determine UNS
17 FN 2: Q12 = +Q10 – +Q11	Calculating the expansion radius RAD

`18 FN 4: Q12 = +Q12 DIV +2`	
`19 FN 1: Q13 = +Q1 + +Q11`	Calculate T–UNS
`20 CYCL DEF 17.0 DISK`	
`21 CYCL DEF 17.1 Z+Q13 M36`	Cycle DISK (see "Cycle 17 DISK" on page 142),
	Depth T–UNS; eroding ON
`22 CYCL DEF 17.2 RAD=Q12 MOD=0`	Expansion radius Q12; expand circularly
`23 L Z+100 R0 FMAX M37`	Retract electrode; eroding OFF
`24 END PGM QDISK MM`	



11

Test run and Program Run

11.1 Graphics

Function

In the program run modes of operation as well as in the Test Run mode, the TNC graphically simulates the machining of the workpiece. Use the BLK FORM ON/OFF soft key to determine if the graphic should be shown or not. Using soft keys, select whether you desire:

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

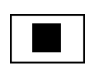
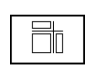
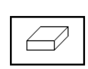
The TNC will not show a graphic if

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

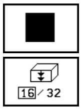
> A graphic simulation is not possible for program sections or programs in which rotary axis movements or a tilted working plane are defined. In this case, the TNC will display an error message.
>
> The TNC graphic does not show a radius oversize DR that has been programmed in the TOOL CALL block.
>
> The TNC can display the graphic only if the ratio of the short side to the long sides of the **BLK FORM** is greater than 1:64!

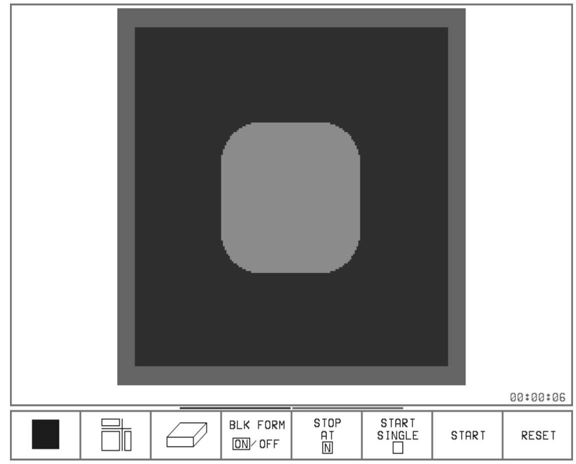
Overview of display modes

The TNC displays the following soft keys in the program run and test run modes of operation:

Display mode	Soft key
Plan view	
Projection in 3 planes	
3-D view	

Plan view





- Press the soft key for plan view.
- Select the number of depth levels (after shifting the soft-key row). You can choose between 16 or 32 shades of depth.

  The deeper the surface, the darker the shade.

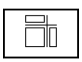
  Plan view is the fastest of the three graphic display modes.

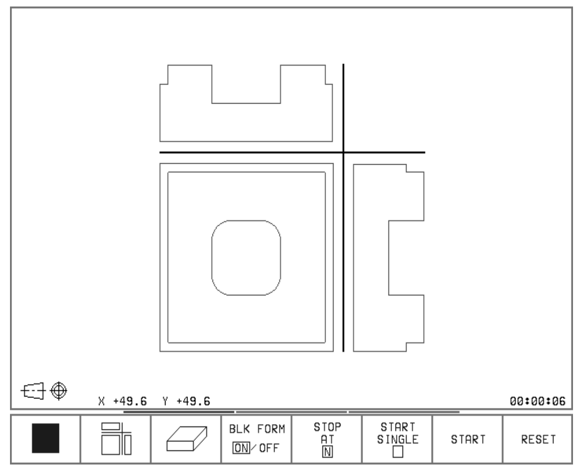
Projection in 3 planes

Similar to a workpiece drawing, the part is displayed with a plan view and two sectional planes. A symbol to the lower left indicates whether the display is in first angle or third angle projection according to ISO 6433 (selected with MP7310).

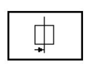
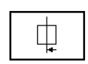
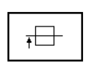
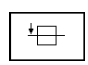
Details can be isolated in this display mode for magnification.

In addition, you can shift the sectional planes with the corresponding soft keys:





- Press the soft key for projection in three planes.
- Shift the soft-key row until the TNC displays the following soft keys:

Function	Soft keys	
Shift the vertical sectional plane to the right or left		
Shift the horizontal sectional plane upwards or downwards		

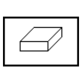
The positions of the sectional planes are visible during shifting.

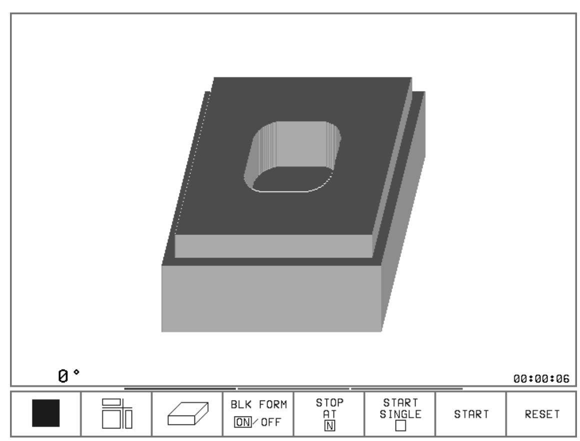
3-D view

The workpiece is displayed in three dimensions, and can be rotated about the vertical axis.

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

In the Test Run mode of operation you can isolate details for magnification.

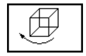
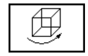




- Press the soft key for 3-D view.

To rotate the 3-D view

Shift the soft-key row until the following soft keys appear:

Function	Soft keys
Rotate the workpiece in 27° steps about the vertical axis	 

The current angle of rotation of the display is shown at the bottom left of the graphic.

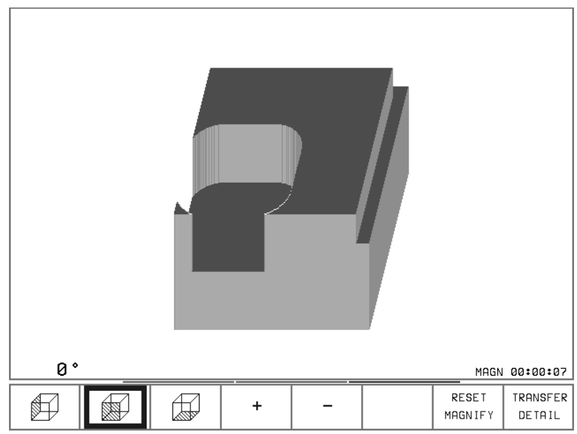
Switch the frame overlay display for the workpiece blank on/off:

SHOW BLK-FORM

OMIT BLK-FORM

- Show the frame overlay with SHOW BLK-FORM
- Omit the frame overlay with OMIT BLK-FORM

Magnifying details

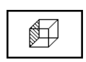
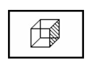
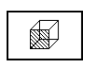
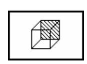
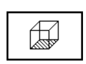
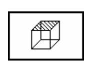


You can magnify details in the Test Run mode in the following display modes:

- Projection in three planes
- 3-D view

The graphic simulation must first have been stopped. A detail magnification is always effective in all display modes.

Shift the soft-key row in the Test Run mode of operation until the following soft keys appear:

Function	Soft keys
Select the left/right workpiece surface	 
Select the front/back workpiece surface	 
Select the top/bottom workpiece surface	 
Shift the sectional plane to reduce or magnify the blank form	− +
Select the isolated detail	TRANSFER DETAIL

To change the detail magnification:

The soft keys are listed in the table above.

- Interrupt the graphic simulation, if necessary.
- Select the workpiece surface with the corresponding soft key (see table).
- To reduce or magnify the blank form, press and hold the MINUS or PLUS soft key, respectively.
- Restart the test run or program run by pressing the START soft key (RESET + START returns the workpiece blank to its original state).

Cursor position during detail magnification

During detail magnification, the TNC displays the coordinates of the axis that is currently being isolated. The coordinates describe the area determined for magnification. To the left of the slash is the smallest coordinate of the detail (MIN point), to the left is the largest (MAX point).

If a graphic display is magnified, this is indicated with **MAGN** at the lower right of the graphics window.

If the workpiece blank cannot be further enlarged or reduced, the TNC displays an error message in the graphics window. To clear the error message, reduce or enlarge the workpiece blank.

Repeating graphic simulation

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

Function	Soft key
Restore workpiece blank to the detail magnification in which it was last shown.	RESET BLK FORM
Reset detail magnification so that the machined workpiece or workpiece blank is displayed as it was programmed with BLK FORM.	RESET MAGNIFY

With the RESET MAGNIFY soft key, you return the displayed workpiece blank to its originally programmed dimensions, even after isolating a detail without TRANSFER DETAIL.

11.2 Test run

Function

In the TEST RUN mode of operation you can simulate programs and program sections to prevent errors from occurring during program run. The TNC checks the programs for the following:

- Geometrical incompatibilities
- Missing data
- Impossible jumps

The following TNC functions can be used in the TEST RUN mode of operation:

- Blockwise test run
- Optional Block Skip
- Functions for graphic simulation

Running a program test

- Select the Test Run mode of operation.
- Choose the program you want to test.
- Press the soft key START. The TNC then tests the program to its end or up to the next programmed interruption.

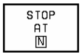
The TNC then displays the following soft keys:

Function	Soft key
Reset program, status and workpiece blank	RESET
Test the entire program	START
Interrupt the test run	STOP
Test each program block individually	START SINGLE
Run a program test up to a certain block	STOP AT N
Run program test with graphics (framing around ON) or without graphics (framing around OFF).	BLK-FORM ON/OFF

Running a program test up to a certain block

If you only want to test the program up to a particular block:

- Choose the program you want to test.

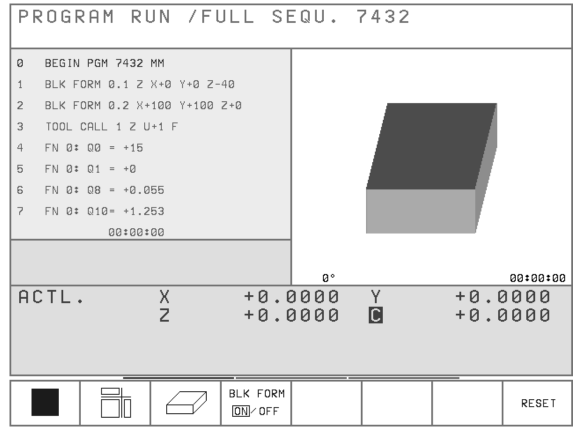


- Press the soft key STOP AT N.
- Enter the block number up to which the TNC should run a program test.
- If the block is located in a different program, enter the PROGRAM.
- If the block number is located within a program section repeat, enter the REPEATS.
- Start the test run with START.

Operating time

The TNC displays the simulation time between the program blocks and the status display.

11.3 Program run



Application

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

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

The following TNC functions can be used in a program run:

- Interrupt program run
- Checking and changing Q parameters
- Functions for graphic simulation

Background programming

During program run it is possible to switch over to the PROGRAMMING AND EDITING mode and enter a new program or edit an existing one while the program being executed continues in the background.

Operating time

The TNC displays the calculated machining time between the program blocks and the status display. The TNC resets the counter for the machining time when you select a new program.

Changing the erosion parameters during program run

The TNC displays the erosion parameters of the current power stage in a line on the screen. You can select the individual erosion parameters with the horizontal arrow keys. Then use the vertical arrow keys to change the parameters settings while the program is being executed. Parameter settings that are changed during program run will not be entered in the erosion table.

> The machine tool builder can inhibit changing of specific erosion parameters. Refer to your machine tool manual.

Running a part program

Preparation

1 Clamp the workpiece to the machine table.

2 Set the datum.

Program Run, Full Sequence

▶ Start the part program with the machine START button.

Program Run, Single Block

▶ Start each block of the part program individually with the machine START button.

Interrupting machining

There are several ways to interrupt a program run:

- Programmed interruptions
- Machine STOP button
- Switching to PROGRAM RUN, SINGLE BLOCK

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

Programmed interruptions

You can program interruptions directly in the part program. The TNC interrupts the program run at a block containing one of the following entries:

- STOP
- Miscellaneous function M0, M2 or M30
- Miscellaneous function M6 (determined by the machine tool builder)

Interrupting or aborting a program by pressing a button

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

0	Interrupt program run.

The ✻ symbol in the status display blinks.
Once you have pressed the Hand soft key, the axes can be traversed manually using the axis-direction keys. To approach the point of interruption once again, use the "Return to contour" function (see "Resuming program run after an interruption" on page 225).

Program run can be aborted with the machine STOP button.

STOP	Abort program run.

The ✻ symbol in the status display goes out.

Interruption of machining by switching to the PROGRAM RUN, SINGLE BLOCK mode of operation

The machining process is interrupted at the end of the current block.

	Select PROGRAM RUN, SINGLE BLOCK.

Mid-program startup (block scan)

If you want to start the program not at the first block but at some other block:

- Test the program in the operating mode TEST RUN up to the desired block.
- Switch to the program mode PROGRAM RUN.
- Start the program at the current block.

The TNC moves the axes towards the contour in a pre-determined sequence (positioning logic). While the TNC is moving the axes, a message is displayed indicating that reapproach is active. You can switch back and forth between the operating modes TEST RUN and PROGRAM RUN as often as desired.

Resuming program run after an interruption

After an interruption you can resume program run at the point where the program was interrupted. M functions that are not evaluated by the NC must first be manually activated.

If program run was interrupted during a fixed cycle, you must restart at the beginning of the cycle. Steps which have already been carried out will then be performed again.

If you have interrupted a called program during program run, the TNC automatically offers the main program when you press the PGM NAME (or PGM MGT) key.

Resuming program run with the START button

It is possible to resume program run with the machine START button if the program was interrupted in one of the following ways:

- The machine STOP button was pressed
- A programmed interruption

Resuming program run after an error

If the error message is not blinking:

- Remove the cause of the error.
- To clear the error message from the screen, press the CE key.
- Restart the program, or resume program run at the place at which it was interrupted.

If the error message is blinking:

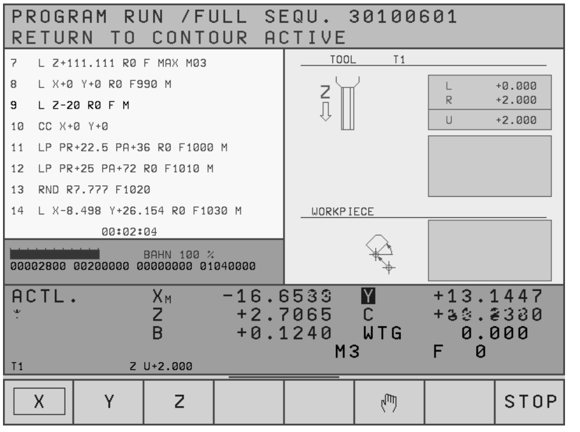
- Switch off the TNC and the machine.
- Remove the cause of the error.
- Start again.

If you cannot correct the error, write down the error message and contact your repair service agency.

Returning to the interruption spot

After interrupting machining with the NC Stop key, you can use the Hand soft key to move the machine axes in the MANUAL mode, e.g. to check the electrode for potential damage. Then you can have the TNC reposition the electrode to the point of the interruption:

- Interrupt program run: Press the NC Stop key, the * symbol in the status display starts blinking.
- Press the Hand soft key to be able to traverse the machine axes.
- Using the axis-direction keys, move the electrode to any position.
- To reapproach the interruption position: Press the RESTORE POSITION key and the TNC activates the ”Return to contour” function (see figure at right).
- Using the soft keys, select the axis to be repositioned and then press NC Start.
- Reposition all of the axes to the interruption point in the same way.
- Resume program run with NC Start.



Resuming program run with the GOTO key

You can interrupt workpiece machining (PROGRAM RUN/FULL SEQUENCE) and move the machine axes manually.

- When a program is resumed by pressing the GOTO key, program blocks that are located before the selected block are ignored!
- After manual positioning the TNC moves the axes to incremental coordinates referenced to the actual position of the tool, and **not** to the last programmed position.

Resetting the counters

To reset the counters of program section repeats after you resume program run:

- Use GOTO 0 to return to the beginning of the program.

If you do **not** wish to reset the counters:

- Use GOTO > 0 to go to a certain block.

Time capture table TIME.W

The time capture table TIME.W in the TNC contains the following columns:

- PS: Power stage number
- ETABLE: Erosion table name
- TOOL: Tool number
- REL.: Erosion time per power stage
- ABS.: Total erosion time
- DATUM: Datum table name
- NR: Datum number
- PROGRAM: Program name
- BLOCK: Block number

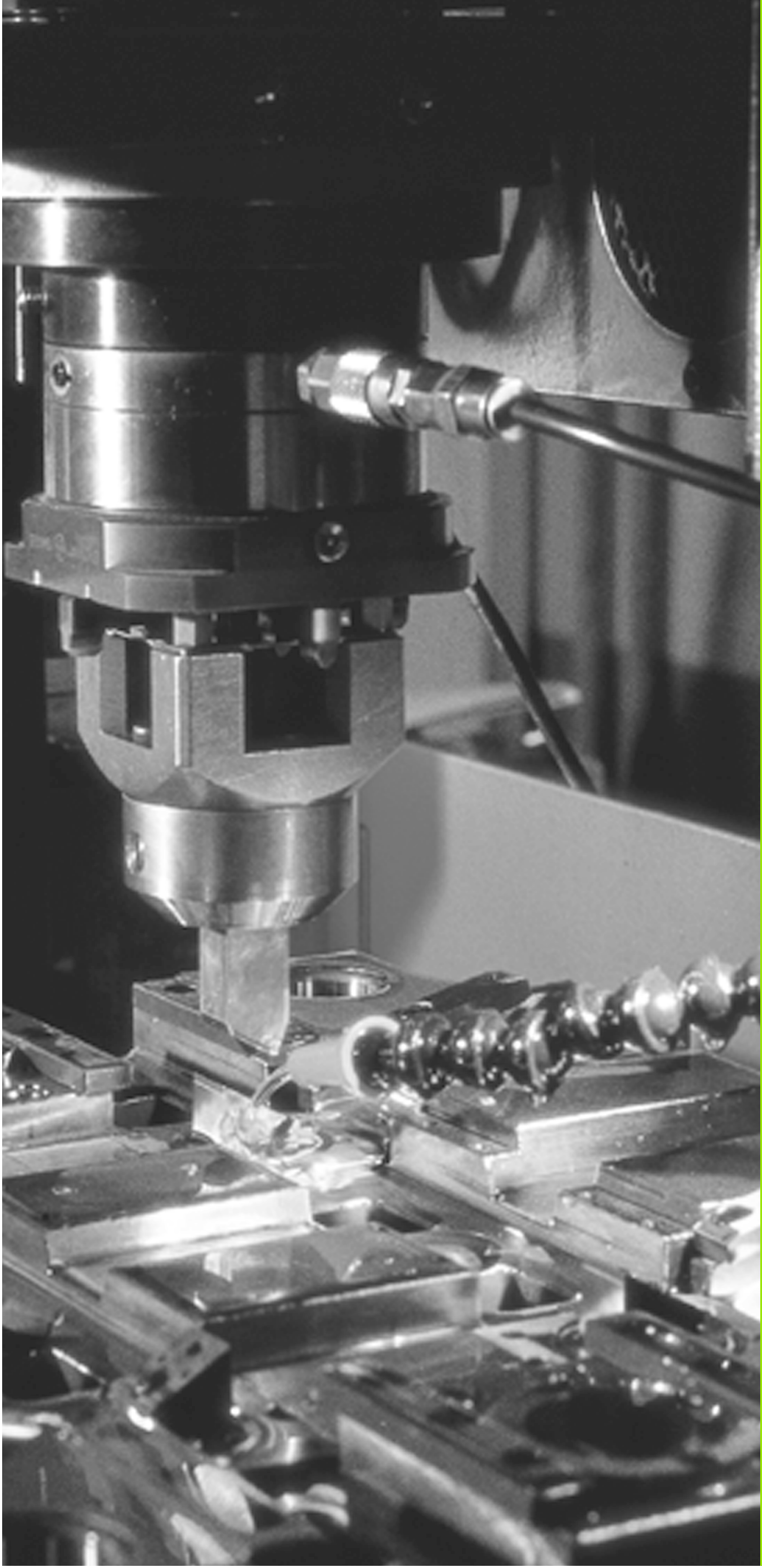
The TNC automatically writes the above data to TIME.W when a new generator setting is transmitted to the PLC during program run. The time table therefore receives as many lines as there are power stages programmed.

To display TIME.W

- Select the operating mode PROGRAMMING AND EDITING.
- Press the PGN NAME key.
- Enter "TIME".

Resetting TIME.W

The TNC automatically overwrites TIME.W when you select a new program in a program run mode of operation or press the RESET soft key.



12

MOD Functions

12.1 MOD functions

Selecting, Changing and Exiting the MOD Functions

▶ Press the toggle key.

▶ Select the desired MOD function with the corresponding soft key.

▶ Use the horizontal arrow keys to change the setting, or enter a value.

▶ Press the END key to exit the MOD function.

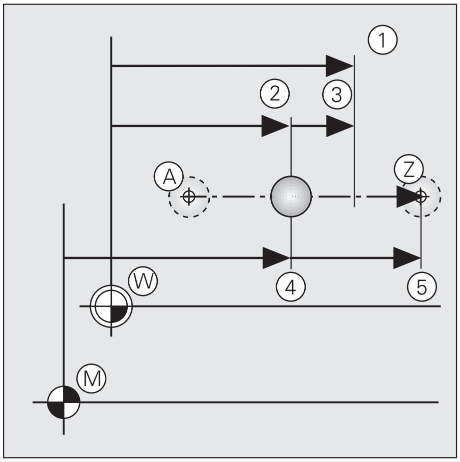
▶ Press the toggle key to return to the mode of operation from which you called MOD.

Overview of MOD functions

The MOD functions provide additional displays and input possibilities. They are selected with soft keys. The following functions are available:

- Position Display Types
- Unit of measurement (mm/inches)
- System information (NC and software numbers)
- Set data interface
- Axis traverse limits
- Machine-specific user parameters
- Enter code number
- Q parameter status in test run or in program run mode of operation

Position Display Types



The positions indicated in the figure are:

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

The TNC position displays can show the following coordinates:

Function	Display
Nominal position: the instantaneous value commanded by the TNC **1**	NOML.
Actual position: the position at which the tool is presently located **2**	ACTL.
Servo lag: difference between nominal and actual positions **3**	LAG
Reference position; the actual position relative to the scale datum **4**	REF
Distance remaining to the programmed position; difference between actual and target positions **5**	DIST.
Nominal position referenced to the transformed coordinate system, such as after a datum shift	NOM.W
Actual position referenced to the transformed coordinate system, such as after a datum shift	ACT.W

Select the desired display type with the **horizontal arrow keys.** It immediately appears in the status field.

Unit of measurement

This MOD function determines whether the coordinates are displayed in millimeters (metric system) or inches.

- To select the metric system (e.g., X = 15.789 mm), set the MOD function CHANGE MM/INCH to MM. The value is displayed to 3 decimal places.
- To select the inch system (e.g., X = 0.6216 inch) set the MOD function CHANGE MM/INCH to INCH. The value is displayed to 4 decimal places.

System Information

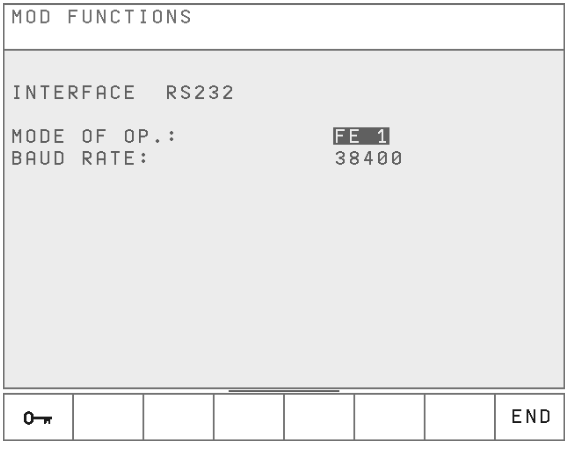
The NC and PLC software numbers appear on the TNC screen after the corresponding MOD functions have been selected. The vacant memory in bytes is displayed directly below them.

Setting the external data interfaces

Two functions are available for setting the external data interfaces:

- BAUD RATE
- RS-232-C INTERFACE

The functions are selected as MOD functions with the vertical arrow keys.



BAUD RATE

Sets the speed of data transfer.

Available baud rates:
110, 150, 300, 600, 1 200, 2 400, 4 800, 9 600, 19 200, 38 400 baud.

To change the baud rate setting:
Press the horizontal arrow keys.

RS-232-C interface

The proper setting depends on the device connected.

Use the ENT key to select the setting.

External device	RS-232-C interface
HEIDENHAIN floppy disk units FE 401 and FE 401 B	FE 1
HEIDENHAIN ME 101 magnetic tape unit, non-HEIDENHAIN devices such as printers, scanners, tape punchers, PC without TNC.EXE	EXT

12.2 External Data Transfer

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

Application examples

- Downloading files into the TNC
- Transferring files from the TNC to external devices
- Printing files
- Remote operation of the TNC

The RS-232-C/V.24 interface is used for these operations.

LSV-2 protocol

The TNC supports the LSV-2 protocol. This allows the control of data transfer or of program run, for example.

Protecting files

The functions PROTECT and UNPROTECT are available for external data transfer (see Chapter 1).

12.3 Menu for External Data Transfer

To select external data transfer

EXT Press the EXT key or

PGM MGT Press the PGM MGT key.

EXT Press the soft key EXT: The selected interface mode and the selected baud rate appear on the screen.

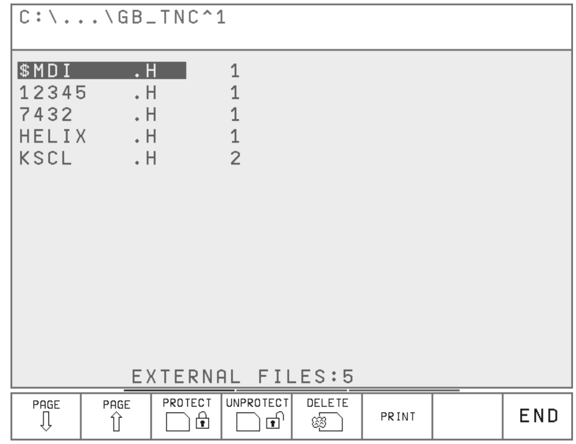
Windows for external data transfer

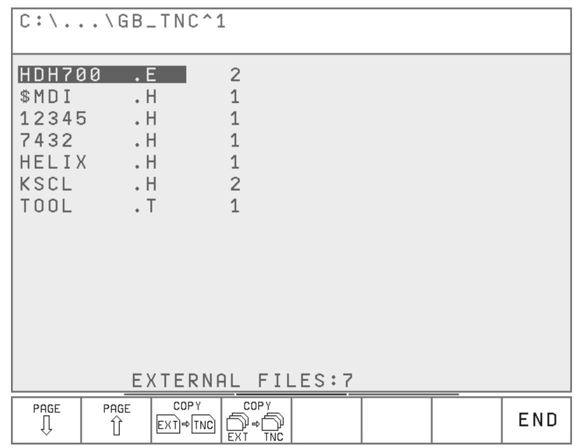
The TNC displays the files in three windows on the screen. You can move from one window to another with the switch-over keys to the left and right of the soft keys.

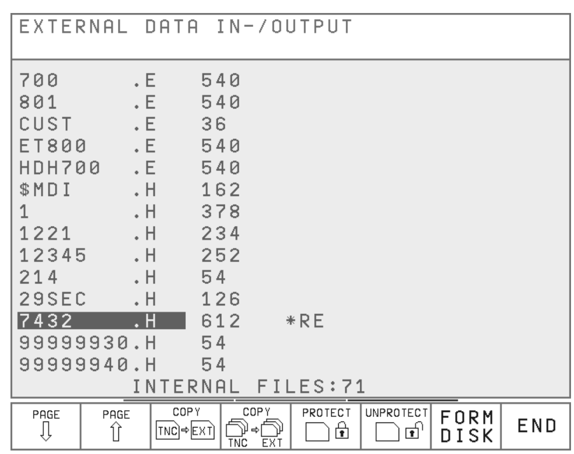
Top window:	All files on the external storage device
Middle window:	NC programs and erosion tables on the external storage device (default setting)
Bottom window:	All files in the TNC memory

Under the list of files the TNC displays whether the files are in the TNC memory (**INTERNAL files**) or whether they are stored on an external device (**EXTERNAL files**).

After that the number of files in the displayed directory are shown.







12.4 Selecting and Transferring Files

Selecting the transfer function

The data transfer functions are selected from the soft-key row.

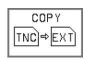
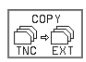
Selecting a file

Select a file with the arrow keys.
The PAGE soft keys are for scrolling up and down in the file directory (the same as in file management).

Transferring files

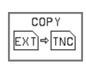
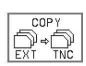
Transferring files from the TNC to an external device

The highlight is on a field stored in the TNC.

Function	Soft key
Transfer the selected file	
Transfer all files	

Transferring files from an external device to the TNC

Use a cursor key to move the highlight to a file that is stored in the external device.

Function	Soft key
Transfer the selected file	
Transfer all files	

Interrupting data transfer

Press the END key or the END soft key to interrupt data transfer.

Transferring files via the PRT output of the FE 401

You can also transfer files via the PRT output of the FE 401 to devices such as a printer:

- Select the file and press the PRINT soft key.

The functions "Transfer all files," "Transfer selected file," and "Transfer directory" are not available in the operating modes FE2 and EXT.

Formatting disks

If you want to save files to a disk, the disk must be formatted. You can format a disk in the FE 401 from the TNC keyboard:

- Press the FORM DISK soft key.
- Enter a name for the disk.
- Press ENT.
  The TNC then formats the disk.

Deleting files

To delete a file on an external device:

- Use the arrow keys to select the file.
- Press the DELETE soft key.

12.5 Software for Data Transfer

Software for data transfer

For transfer of files to and from the TNC, we recommend using one the HEIDENHAIN TNCremo data transfer software products for data transfer, such as TNCremo or TNCremoNT. With TNCremo/ TNCremoNT, data transfer is possible with all HEIDENHAIN controls via serial interface.

> Please contact your HEIDENHAIN agent if you would like to receive the TNCremo or TNCremoNT data transfer software for a nominal fee.

System requirements for TNCremo:

- AT personal computer or compatible system
- Operating system MS-DOS/PC-DOS 3.00 or later, Windows 3.1, Windows for Workgroups 3.11, Windows NT 3.51, OS/2
- 640 KB working memory
- 1 MB free memory space on your hard disk
- One free serial interface
- A Microsoft-compatible mouse (for ease of operation, not essential)

System requirements for TNCremoNT:

- PC with 486 processor or higher
- Operating system Windows 95, Windows 98, Windows NT 4.0
- 16 MB working memory
- 5 MB free memory space on your hard disk
- One free serial interface or connection to the TCP/IP network on TNCs with Ethernet card

Installation under Windows

- Start the SETUP.EXE installation program in the file manager (Explorer).
- Follow the instructions of the setup program

Starting TNCremo under Windows 3.1, 3.11 and NT 3.51

Windows 3.1, 3.11, NT 3.51:

- Double-click on the icon in the program group HEIDENHAIN Applications

When you start TNCremo for the first time, you will be asked for the type of control you have connected, the interface (COM1 or COM2) and the data transfer speed. Enter the necessary information.

Starting TNCremoNT under Windows 95, Windows 98 and NT 4.0

- Click on <Start>, <Programs>, <HEIDENHAIN Applications>, <TNCremoNT>

When you start TNCremoNT for the first time, TNCremoNT automatically tries to set up a connection with the TNC.

Data transfer between the TNC and TNCremo

Ensure that:

- The TNC is connected to the correct serial port on your PC.
- The data transfer speed set on the TNC is the same as that set on TNCremo.

Once you have started TNCremo, you will see a list of all of the files that are stored in the active directory on the left side of the main window **1** . Using the menu items <Directory>, <Change>, you can change the active directory or select another directory on your PC.

If you want to control data transfer from the PC, establish the connection with your PC in the following way:

- Select <Connect>, <Link (LSV2)>. TNCremo now receives the file and directory structure from the TNC and displays this at the bottom left of the main window **2**.
- To transfer a file from the TNC to the PC, select the file in the TNC window (highlighted with a mouse click) and activate the functions <File> <Transfer>.
- To transfer a file from the PC to the TNC, select the file in the PC window (highlighted with a mouse click) and activate the functions <File> <Transfer>.

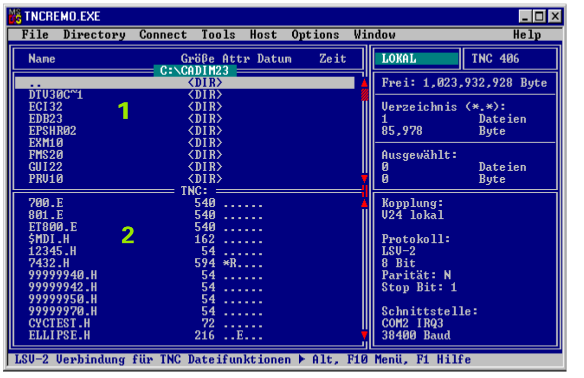
If you want to control data transfer from the TNC, establish the connection with your PC in the following way:

- Select <Connect>, <File server (FE)>. TNCremo is now in server mode. It can receive data from the TNC and send data to the TNC.
- You can now call the file management functions on the TNC by pressing the key PGM MGT, in order to transfer the desired files.

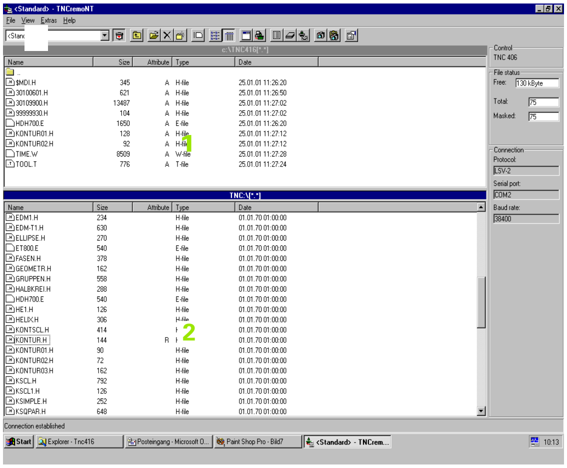
End TNCremo

Select the menu items <File>, <Exit>, or press the key combination ALT+X.

Refer also to the TNCremo help texts where all of the functions are explained in more detail.



Data transfer between the TNC and TNCremoNT



Ensure that:

- The TNC is connected to the correct serial port on your PC.
- The TNCremoNT operating mode is set to LSV2.
- The data transfer speed set on the TNC is the same as that set on TNCremoNT.

Once you have started TNCremoNT, you will see a list of all of the files that are stored in the active directory on the upper section of the main window **1** . Using the menu items <File>, <Change directory>, you can change the active directory or select another directory on your PC.

If you want to control data transfer from the PC, establish the connection with your PC in the following way:

- ▶ Select <File>, <Setup connection>. TNCremoNT now receives the file and directory structure from the TNC and displays this at the bottom left of the main window **2**.
- ▶ To transfer a file from the TNC to the PC, select the file in the TNC window with a mouse click and drag and drop the highlighted file into the PC window **1**.
- ▶ To transfer a file from the PC to the TNC, select the file in the PC window with a mouse click and drag and drop the highlighted file into the PC window **2**.

If you want to control data transfer from the TNC, establish the connection with your PC in the following way:

- ▶ Select <Extras>, <TNCserver>. TNCremoNT is now in server mode. It can receive data from the TNC and send data to the TNC.
- ▶ You can now call the file management functions on the TNC by pressing the key PGM MGT, in order to transfer the desired files.

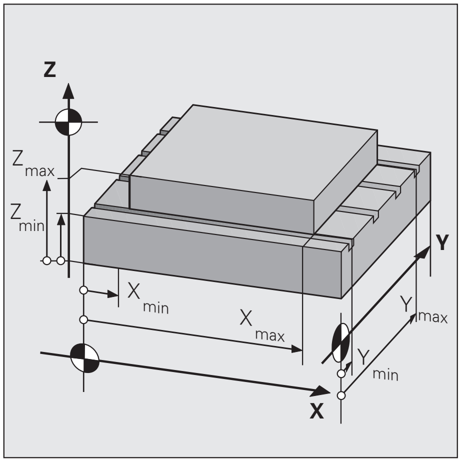
End TNCremoNT

Select the menu items <File>, <Exit>.

> Refer also to the TNCremoNT help texts where all of the functions are explained in more detail.

12.6 Enter Axis Traverse Limits

Introduction



The AXIS LIMIT mod function allows you to set limits to axis traverse within the machine's actual working envelope.

Example application:

To protect an indexing fixture against tool collision.

The maximum range of traverse of the machine tool is defined by software limit switches. This range can be additionally limited with the AXIS LIMIT mod function. With this function you can enter the maximum and minimum traverse positions for each axis, referenced to the machine datum.

Working without additional traverse limits

To allow a machine axis to use its full range of traverse, enter the maximum traverse of the TNC (+/– 30 000 mm) as the AXIS LIMIT.

To find and enter the maximum traverse:

Select POSITION DISPLAY REF

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

Write down the values, including the algebraic sign.

MOD — MOD functions

AXIS LIMIT — Press the AXIS LIMIT soft key.

and enter the values that you wrote down as LIMITS in the corresponding axes

END — Exit the MOD function.

- The tool radius is not automatically compensated in the axis traverse limit value.
- The traverse range limits and software limit switches become active as soon as the reference points are traversed.
- The TNC checks whether the negative limit is less than the positive limit in each axis.
- You can also transfer the reference points directly, using actual position capture.
- You can re-establish the last stored value with the NO ENT key.
- You can re-establish the axis limits stored in the machine parameters with the TRANSFER FROM MP soft key.

12.7 Machine-Specific User Parameters

Function

The machine tool builder can assign functions to up to 16 user parameters. Refer to your machine tool manual.

12.8 Code Number

Function

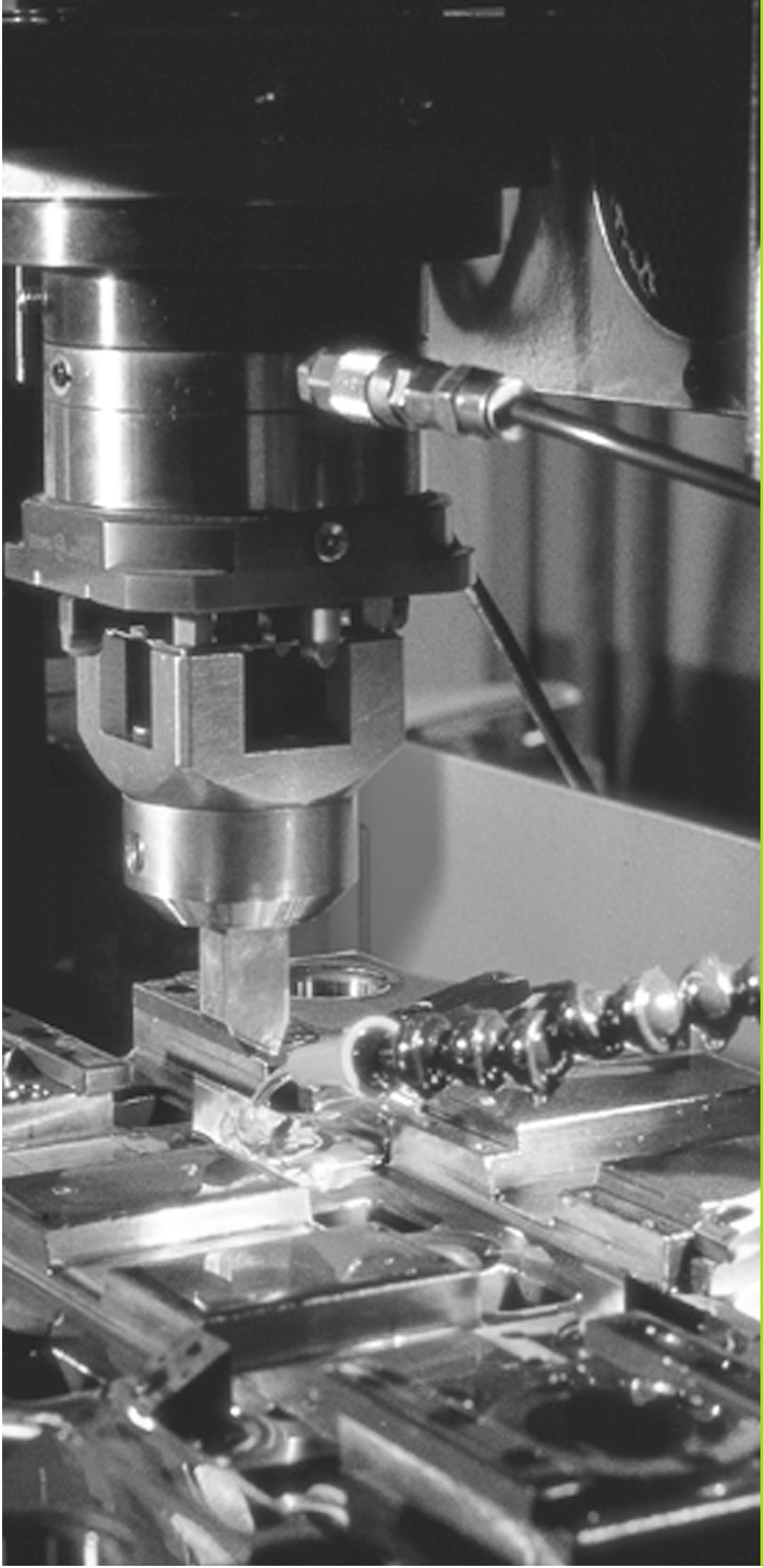
If you want to change the user parameters, you must first enter the code number 123 (see "General User Parameters" on page 246).

Enter the code number after selecting the corresponding MOD function in the dialog field. The TNC displays one asterisk for each digit you enter.

12.9 Q Parameter Status Display

Function

With the Q-PAR soft key you can check and, if necessary, change the currently defined Q parameters while the TNC is running a program test or part program (see ”Checking and Changing Q Parameters” on page 196).



13

Tables and Overviews

13.1 General User Parameters

General user parameters are machine parameters affecting TNC settings that the user may want to change in accordance with his requirements. Some examples of user parameters are:

- Dialog language
- Interface behavior
- Traversing speeds
- Effect of overrides

Entering machine parameters

Machine parameters can be programmed as decimal numbers.

Some machine parameters have more than one function. The input value for these machine parameters is the sum of the individual values. For these machine parameters the individual values are preceded by a plus sign.

Selecting the General User Parameters

To access the general user parameters, enter code number 123 in the MOD functions.

The MOD functions also include machine-specific user parameters (USER PARAMETERS).

Machining feed rate				
Default feed rate for positioning (when no feed rate is programmed)	**MP1090** **0** to **30 000** [mm/min]			
Maximum circular feed rate in Cycle 17 DISK		**Modes**	**Operating mode**	**Value**
	MP1092 **MP1093**	0 and 4 0 and 4	Eroding Free run	**0** to **30 000** [mm/min] **0** to **30 000** [mm/min]
	MP1094 **MP1095**	1 and 5 1 and 5	Eroding Free run	**0** to **30 000** [mm/min] **0** to **30 000** [mm/min]
	MP1096 **MP1097**	2 and 6 2 and 6	Eroding Free run	**0** to **30 000** [mm/min] **0** to **30 000** [mm/min]

Eroding	
Value for Q157 with TOOL CALL or EL-CALL block	**MP2040** **0.1** to **10**
Advanced stop distance after short circuit or CYCL STOP After a short circuit or CYCL STOP block, the TNC moves the electrode back towards the workpiece, but stops at a certain distance from the workpiece. This distance is entered in MP2050.	**MP2050** **0** to **2** [mm]
Advanced stop distance after flushing After flushing the electrode gap, the TNC moves the electrode back towards the workpiece, but stops at a certain distance from the workpiece. This distance is entered in MP2051.	**MP2051** **0** to **2** [mm]
Advanced stop for oscillator signal At the end of a programmed eroding time, the TNC moves the electrode back towards the workpiece. When the electrode reaches the distance from the workpiece that was entered in MP2052, the TNC reactivates the oscillator signal of the generator. This ensures that the TNC always receives the correct analog gap signal when switching from positioning to gap control.	**MP2052** **0** to **2** [mm]
Rotational speed of the C axis with M3/M4 When M3 or M4 are programmed, the C axis rotates at the speed entered in this user parameter.	**MP2090** **0** to **100** [rpm]
Duration of the free run signal after eroding The duration of the free run signal when the programmed eroding step is completed is determined in this user parameter.	**MP2110** **0.1** to **99.9** [s]
Arc detection The TNC recognizes an arc that exists as long as defined in this user parameter.	**MP2120** **1** to **99.9** [s]
Free-run feed rate (only with gap control via gap signal) If the voltage at the analog input is greater than the threshold for the free-run feed rate, the TNC positions the electrode at the feed rate entered in this machine parameter.	**MP2141** **0** to **3000** [mm/min]

Eroding	
Gap-control feed rate (only with gap control via gap signal) If the voltage at the analog input is less than the threshold for the free-run feed rate, the TNC positions the electrode at the feed rate entered in this machine parameter, multiplied by a factor from the PLC.	**MP2142** **1** to **99.9** [mm/min]

External data transfer	
Control character for end of file	**MP5010** Control character for end of text (e.g., MP 5010=3: EXT): **ASCII character** Do not send control character for end of text: **0**
Control character for end of transmission	**MP5011** Control character for end of text (e.g., MP 5011=4: EOT): **ASCII character** Do not send control character for end of transmission: **0**
Adapt TNC interface to an 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) control character not permitted: **+2** Transmission stop through RTS active: **+4** Transmission stop through RTS inactive: **+0** Transmission stop through DC3 active: **+8** Transmission stop through DC3 inactive: **+0** Character parity even: **+0** Character parity odd: **+16** Character parity not desired: **+0** Character parity desired: **+32** 2 stop bits: **+64** 1 stop bit: **+128** Example: Use the following setting to adjust the TNC interface to an external non-HEIDENHAIN device: 8 data bits, any BCC, transmission stop through DC3, even character parity, character parity desired, 2 stop bits Input value: 1+0+8+0+32+64 = 105 (input value for MP5020)
Parity setting for LSV-2 protocol	**MP5100** No parity: **0** Even parity: **1** Odd parity: **2**

External data transfer	
Baud rate for RS-422 interface of the PLC	**MP5200** 9600: **0** 38400: **1**
Check sequence of blocks with external data transfer	**MP5990** Check sequence of blocks with external data transfer: **0** No check: **1**

Parameters for probing with the TCH PROBE function	
Number of times the probing process is repeated for probing a workpiece	**MP6100** **0** to **5**
Maximum difference between results from probing a workpiece several times The TNC aborts probing and generates an error message if the difference between results exceeds the value entered in MP6110.	**MP6110** **0** to **2** [mm]
Probe feed rate	**MP6120** **80** to **3 000** [mm/min]
Maximum traverse to first probe point The TNC aborts probing and generates an error message if the electrode does not reach the workpiece within the defined measuring range.	**MP6130** **0** to **30 000** [mm]
Distance by which the electrode is retracted when probing manually If 0 is entered, the electrode is always retracted to the starting point.	**MP6140** **0** to **30 000** [mm]
Retraction time after the end of electrode contact when probing manually	**MP6141** **10** to **400** [ms]
Rapid traverse for probing After probing, the TNC retracts the electrode at the speed defined in this parameter.	**MP6150** **1** to **30 000** [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**

TNC displays, TNC editor	
Disabling file types MP7224 is bit-coded. If bit 2 is set, new tool tables cannot be created. Existent tool tables may still be edited, but the changes will be ignored in program run.	**MP7224** Not disabled: **0** File type disabled for ■ Bit 0 vacant ■ Bit 1 vacant ■ Bit 2 tool table: 4 ■ All other bits are vacant
Dialog language	**MP7230** Input value: 0 to 3 Languages for software 280620 ■ English = 0 ■ German = 1 ■ French = 2 ■ Italian = 3 Languages for software 280621 ■ English = 0 ■ German = 1 ■ Swedish = 2 ■ Finnish = 3 Languages for software 280622 ■ English = 0 ■ German = 1 ■ Czech = 2 ■ Reserved = 3
Protect OEM cycles in the TNC memory	**MP7240** Protect OEM cycles: **0** Do not protect OEM cycles: **1**
Inhibit EL-CALL and WP-CALL soft keys	**MP7241** Do not display oft keys: **0** Display oft keys: **1**
Number of pockets in the tool magazine	**MP7261** **0** to **999**
Permit multiple assignment of pocket numbers Aside from pocket number 0, you can prevent a pocket from being assigned more than once.	**MP7265** Assign pocket number only once: **1** Assign pocket number several times: **0**
Display feed rate	**MP7271** Display feed rate F: **0** Do not display feed rate F: **1**

TNC displays, TNC editor	
Display of program blocks during test run	**MP7273** Do not display program blocks: **0** Display program blocks: **1**
Decimal character	**MP7280** The decimal character is a comma: **0** The decimal character is a point: **1**
Display step for the X axis	**MP7290.0** 0.0001 mm, 0.00001 inch: **0** 0.0005 mm, 0.00002 inch: **1** 0.001 mm, 0.0001 inch: **2** 0.005 mm, 0.0002 inch: **3** 0.01 mm, 0.001 inch: **4** 0.05 mm, 0.002 inch: **5** 0.1 mm, 0.01 inch: **6**
Display step for the Y axis	**MP7290.1** Input range see MP7290.0
Display step for the Z axis	**MP7290.2** Input range see MP7290.0
Display step for the IVth axis	**MP7290.3** Input range see MP7290.0
Display step for the 5th axis	**MP7290.4** Input range see MP7290.0
Reset Q parameters and status display	**MP7300** Do not reset: **+0** Reset with M02, M30 and END PGM: **+1** Do not reset Q parameters when selecting a program or pressing the RESET soft key: **+2** Do not reset tool data when selecting a program or pressing the RESET soft key: **+4**
Graphic display mode	**MP7310** Projection in three planes according to ISO 6433, part 1, projection method 1: **+0** Projection in three planes according to ISO 6433, part 2, projection method 1: **+1**
	Do not rotate coordinate system for graphic display: **+0** Rotate coordinate system for graphic display by 90°: **+2**
Graphic simulation without programmed tool: Tool radius	**MP7315** **0.0000** to **9 999.999** [mm]
Graphic simulation without programmed tool: Penetration depth	**MP7316** **0.0000** to **9 999.999** [mm]
Effect of Cycle 11 SCALING FACTOR	**MP7410** SCALING FACTOR effective in 3 axes: **0** SCALING FACTOR effective in the working plane only: **1**

TNC displays, TNC editor	
Effect of axis IV in the datum table	**MP7411** IVth coordinate with datum from table rotates coordinate system and shifts in C: **0** IVth coordinate with datum from table shifts in C (no rotation): **1**
Effect of CYCL CALL after CYCL DEF 12 PGM CALL	**MP7412** The program defined as a cycle is executed without display of NC blocks; local Q parameters are stored: **0** The program defined as a cycle is executed with display of NC blocks; local Q parameters are not stored: **1** **Does not apply as of NC 28612x-04/28062x-10.**
Behavior of M functions	**MP7440** Program stop with M06: **+0** No program stop with M06: **+1**
	No cycle call with M89: **+0** Modal cycle call with M89: **+1**
Maximum permissible angle of directional change for constant contouring speed (effective for corners with R0 and for all inside corners).	**MP7460** **0.0000** to **179.999** [°]
Monitoring limit switches in the TEST RUN mode of operation	**MP7491** Monitoring limit switches active: **0** Monitoring limit switches not active: **1**

Override behavior	
Set overrides	**MP7620** Feed rate override, if rapid traverse key is pressed in program run mode Override effective: **+1** Override not effective: **+0**
	Steps for overrides 2% steps: **+0** 1% steps: **+2**
	Feed rate override, if rapid traverse and external direction axis direction button pressed Override effective: **+4** Override not effective: **+0**

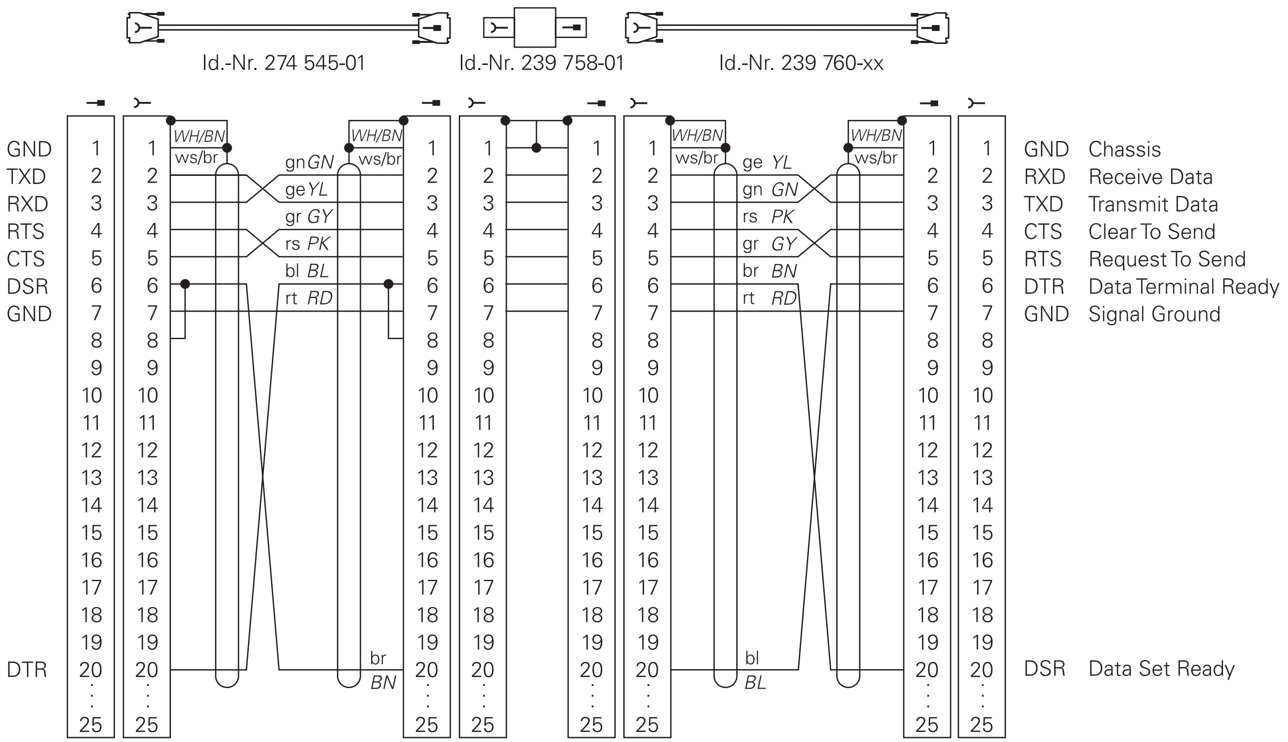
Electronic handwheels	
Set interpolation error	**MP7670.0** Slow handwheel interpolation error: **0...10** **MP7670.1** Medium handwheel interpolation error: **0...10** **MP7670.2** Fast handwheel interpolation error: **0...10**

Electronic handwheels	
Feed rate of the direction keys on the handwheel in percent compared to the machine axis direction buttons on the operating panel	**MP7671.0** Slow feed rate: **0...10** **MP7671.1** Medium feed rate: **0...10** **MP7671.2** Fast feed rate: **0...10**

13.2 Pin Layout and Connecting Cable for the Data Interfaces

RS-232-C/V.24 Interface HEIDENHAIN devices

External device | **HEIDENHAIN standard cable 1 m** | **RS-422 Adapter block** | **HEIDENHAIN connecting cable max. 17 m** | **X21 TNC**

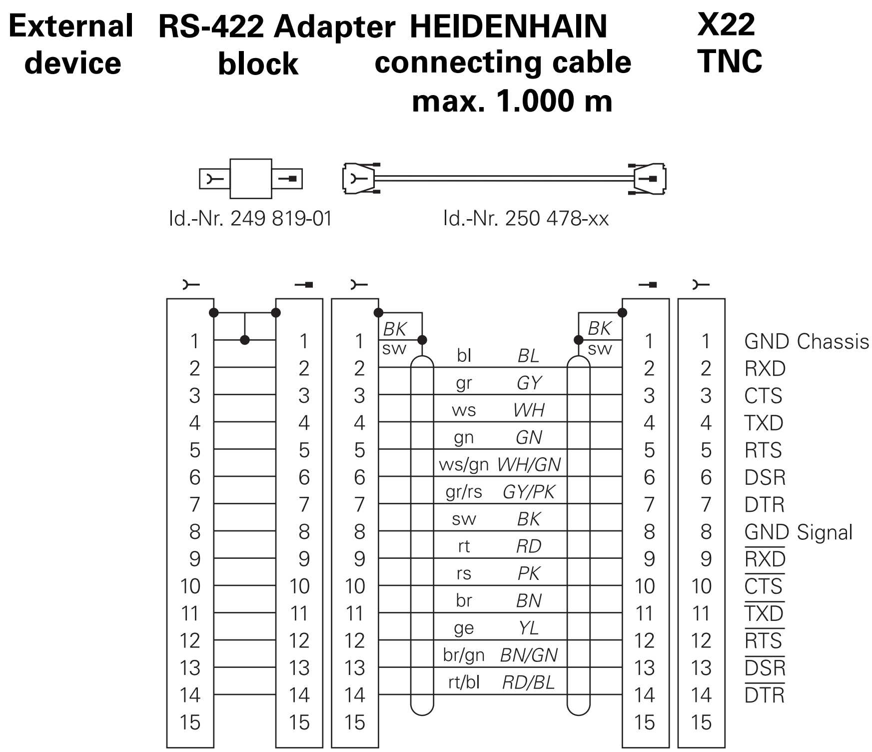


- The connector pin layout on the adapter block differs from that on the TNC logic unit (X21).
- The connector pin layout of a non-HEIDENHAIN device may differ considerably from that on a HEIDENHAIN device.
  This often depends on the unit and type of data transfer. The figure above shows the connector pin layout on the adapter block.

RS-422/V.11 Interface

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

> The pin layouts on the TNC logic unit (X22) and on the adapter block are identical.



13.3 Preparing the Devices for Data Transfer

HEIDENHAIN devices

HEIDENHAIN devices (FE floppy disk unit and ME magnetic tape unit) are already adapted to the TNC. They can be used for data transfer without further adjustments.

Example: FE 401 floppy disk unit

- Connect the power cable to the FE.
- Connect the FE and TNC with the data interface cable.
- Switch on the FE.
- Insert a disk in the upper drive.
- Format the disk if necessary.
- Set data interface (see "Setting the external data interfaces" on page 232).
- Transfer the data.

- The memory capacity of a floppy disk is given in sectors.
- The baud rate can be set on the FE 401.

Non-HEIDENHAIN devices

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

To adapt a non-HEIDENHAIN device to the TNC:

- PC: adapt the software.
- Printer: Set the DIP switches.

To adapt the TNC to a non-HEIDENHAIN device:

Set the user parameters:

- 5010 to 5020 for EXT.

13.4 Technical Information

The TNC 406/416	
Description	Contouring control for ram EDM machines with up to 5 axes
Components	■ Logic unit ■ Keyboard ■ CRT ■ Flat screen (only TNC 416)
Data interfaces	■ RS-232-C / V.24 ■ RS-422 / V.11 ■ Expanded data interface with LSV-2 protocol for remote operation of the TNC through the data interface with the HEIDENHAIN software TNCremo
Simultaneous axis control for contour elements	■ Straight lines: up to 3 axes ■ Circles: up to 2 axes ■ Helix with C axis interpolation
Background programming	One part program can be edited while the TNC runs another program
Graphics	■ Test run graphics
File types	■ HEIDENHAIN conversational programming ■ Erosion tables ■ Tool tables ■ Datum tables
Program memory	Battery buffered for up to 100 files Capacity approximately 10 000 blocks (TNC 406) or 20 000 blocks (TNC 416)

TNC Specifications	
Block processing time	15 ms/block (4 000 blocks/min)
Control loop cycle time	■ TNC 406/416 switchable (2 ms or 4 ms; MP 1700)
Data transfer rate	Max. 38 400 baud
Ambient temperature	■ Operation: 0° C to +45° C (32° to 113° F) ■ Storage: –30°C to +70°C (–22° F to 158° F)
Traverse range	Maximum ± 30 m (1 181 inches)
Traversing speed	Maximum 30 m/min (1 181 ipm)
Input range	To 1 µm (0.0001 inches) or 0.001°
Control precision	1/16 µm

Programmable functions	
Contour elements	■ Straight line ■ Chamfer ■ Circular path ■ Circle center ■ Circle radius ■ Tangentially connecting circle ■ Corner rounding ■ Straight lines and circular arcs for contour approach and departure
Program jumps	■ Subprogram ■ Program section repeat ■ Program as subprogram
Fixed cycles	■ Cycle GENERATOR ■ Erosion Cycles
Coordinate transformations	■ Datum shift ■ Mirror image ■ Rotation ■ Scaling factor
Touch probe function	Touch probe functions for setting datums and for automatic workpiece measurement
Mathematical functions	■ Basic arithmetic +, –, x and / ■ Trigonometry sin, cos, tan, arcsin, arccos, arctan ■ Square root and root sum of squares ■ Logical comparisons (greater than, less than, equal to, not equal to)

Electronic handwheels	
HR 130	For panel mounting
HR 410	Portable version with cable transmission. Includes axis address keys, actual position capture key, 3 keys for selecting the traversing speed, direction keys, machine functions, rapid traverse key, safety switch, emergency stop button.

13.5 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

Some of the more frequent TNC error messages are explained in the following list.
An error message that contains a program block number was caused by an error in the indicated block or in the preceding block.
To clear the TNC error message, first correct the error and then press the CE key.

Error messages that are displayed in the screen center are generated by the TNC. Error messages that appear in the upper screen window for the operating modes are defined by the machine tool builder. Refer to your machine tool manual.

TNC error messages during programming

TNC error messages	Procedure
Further program entry impossible	Erase some old files to make room for new ones.
Entry value incorrect	■ Enter a correct label number. ■ Press the correct key.
Ext. in-/output not ready	Connect the external device properly.
Label number already assigned	A given label number can only be entered once in a program.
Jump to label 0 not permitted	Do not program CALL LBL 0.

TNC error messages during test run and program run

TNC error messages	Procedure
Selected block not addressed	Before a test run or program run, you must enter GOTO 0.
Probed value inaccurate	The difference between individual results from probing the workpiece several times exceeds the maximum allowable difference set in MP6110.
Arithmetical error	You have calculated with non-permissible values ■ Define values within the range limits. ■ Choose probe positions for the probing electrode that are farther apart. ■ All calculations must be mathematically possible.

TNC error messages	Procedure
Path offset wrongly ended	Do not cancel electrode radius compensation in a block with a circular path.
Path offset wrongly started	■ Use the same radius compensation before and after a RND and CHF block. ■ Do not begin electrode radius compensation in a block with a circular path.
CYCL incomplete	■ Define the cycles with all data in the proper sequence. ■ Do not call the coordinate transformation cycles. ■ Before calling a cycle, define Cycle 12 PGM CALL.
BLK FORM definition incorrect	■ Program the MIN and MAX points according to the instructions. ■ Choose a ratio of sides that is less than 64:1. ■ If you call another program (PGM CALL), copy the BLK FORM to the main program.
Axis double programmed	Each axis can have only one value for position coordinates.
Plane wrongly defined	■ Do not change the electrode axis while a basic rotation is active. ■ Correctly define the main axes for a circular arc. ■ Define both main axes for CC.
Wrong axis programmed	■ Do not attempt to program locked axes. ■ Do not mirror rotary axes. ■ Enter a positive chamfer length.
Chamfer not permitted	A chamfer block must be located between two straight-line blocks with identical radius compensation.
No editing of running program	A program cannot be edited while it is being transmitted or executed.
Gross positioning error	The TNC monitors positions and movements. If the actual position deviates excessively from the nominal position, this blinking error message is displayed. To correct the error, do a "warm start" by holding down the END key for a few seconds.
Circle end position incorrect	■ Enter complete information for connecting arc. ■ Enter end points that lie on the circular path.
Label number not found	Only call label numbers that have been set.
PGM section cannot be shown	■ Enter a smaller electrode radius. ■ Movements in a rotary axis cannot be graphically simulated. ■ Enter an electrode axis for simulation that is the same as the axis in the BLK FORM.
Rounding-off undefined	Enter tangentially connecting arcs and rounding arcs correctly.
Rounding radius too large	Rounding arcs must fit between contour elements.
Key non-functional	This message always appears when you press a key that is not needed for the current dialog.

TNC error messages	Procedure
Program start undefined	■ Begin the program only with a TOOL DEF block. ■ Do not resume an interrupted program at a block with a tangential arc or if a previously defined pole is needed.
Tool radius too large	Enter an electrode radius that ■ lies within the given limits ■ permits the contour elements to be calculated and machined.
Angle reference missing	■ Complete your definition of the arc and its end points. ■ If you enter polar coordinates, define the polar angle correctly.
Excessive subprogramming	■ Conclude all subprograms with LBL0. ■ Program CALL LBL for subprograms without REP. ■ Program CALL LBL for program section repeats to include the repetitions (REP). ■ Subprograms cannot call themselves. ■ Subprograms cannot be nested more than 8 levels. ■ Main programs cannot be nested as subprograms in more than 4 levels.

Symbole

Overview of Miscellaneous Functions

Miscellaneous functions with predetermined effect:

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

M	Effect	Effective at block - start	end	Page
M00	Stop program run		■	page 122
M02	Stop program/Clear status display (depending on machine parameter)/Go to block 1		■	page 122
M03	Free rotation of the C axis (direction of rotation set by the machine builder)	■		page 122
M04	Free rotation of the C axis (direction of rotation set by the machine builder)	■		
M05	Stop free rotation of the C axis		■	
M06	Electrode change / Stop program run (dependent on machine parameter 7440)		■	page 122
M08	Flushing ON	■		page 122
M09	Flushing OFF		■	
M13	Functionality of M03 + M08	■		page 122
M14	Functionality of M04 + M08	■		
M30	Same function as M02		■	page 122
M36	Eroding ON / Gap control ON	■		page 122
M37	Eroding OFF / Gap control OFF	■		
M38	Transfer coordinates from datum table 0.D into NC program	■		page 157
M39	Transfer Q parameters from an NC program into the datum table 0.D	■		
M89	Vacant miscellaneous function **or** Cycle call, modally effective (depending on machine parameter MP7440)	■		
M90	Reserved		■	
M91	Within the positioning block: Coordinates are referenced to machine datum	■		page 125
M92	Within the positioning block: Coordinates are referenced to position defined by machine tool builder, such as tool change position	■		page 125
M93	Within the erosion block: Retract the electrode at the end of block and return to the starting point of the machining operation	■		page 125
M94	Reserved	■		
M95	Reserved		■	
M96	Reserved		■	
M97	Machine small contour steps		■	page 123

M	Effect	Effective at block - start	end	Page
M98	Machine open contours completely		■	page 124
M99	Blockwise cycle call		■	
M108	Transfer coordinates from tool table TOOL.T into NC program	■		
M109	Transfer Q parameters from an NC program into the tool table TOOL.T	■		

HEIDENHAIN

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

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

www.heidenhain.de

Ve 00
291 016-24 · 4/2001 · pdf · Subject to change without notice