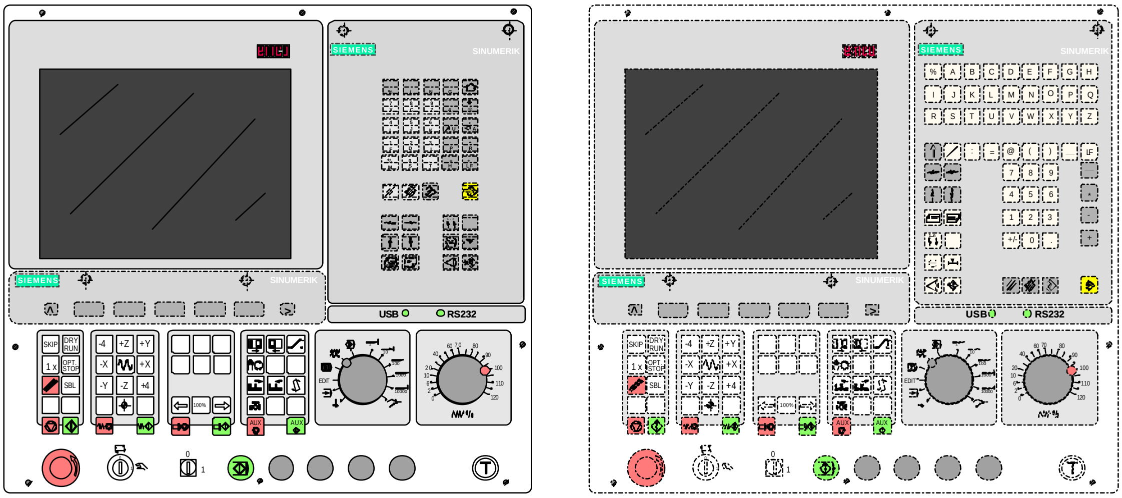
EMCO WinNC SINUMERIK 810/820 M

Software Description/ Software Version from 13.70



Software Description
EMCO WinNC SINUMERIK 810/820 M

Ref.No. EN 1803 Edition J2003-10

EMCO Maier Ges.m.b.H.

P.O. Box 131

A-5400 Hallein-Taxach/Austria

Phone ++43-(0)62 45-891-0

Fax ++43-(0)62 45-869 65

Internet: www.emco.at

E-Mail: service@emco.co.at

innovative machine tools
industrial training systems

Preface

The EMCO WinNC SINUMNERIK 810/820 M Milling Software is part of the EMCO training concept on PC-basis.

This concept aims at learning the operation and programming of a certain machine control on the PC.

The milling machines of the EMCO PC MILL und CONCEPT MILL series can be directly controlled via PC by means of the EMCO WinNC for the EMCO MILL.

The operation is rendered very easy by the use of a digitizer or the control keyboard with TFT flat panel display (optional accessory), and it is didactically especially valuable since it remains very close to the original control.

This manual does not include the whole functionality of the control software SINUMNERIK 810/820 M Milling, however emphasis was laid on the simple and clear illustration of the most important functions so as to achieve a most comprehensive learning success.

In case any questions or proposals for improving this manual should arise, please contact us directly:

EMCO MAIER Gesellschaft m. b. H.
Department for technical documentation
A-5400 Hallein, Austria

All rights reserved, reproduction only by authorization of Messrs. EMCO MAIER
© EMCO MAIER Gesellschaft m.b.H., Hallein 2003

Contents

A: Key Description

B: Basics

C: Operating Sequences

D: Programming

E: @-Codes

G: Survey Pages

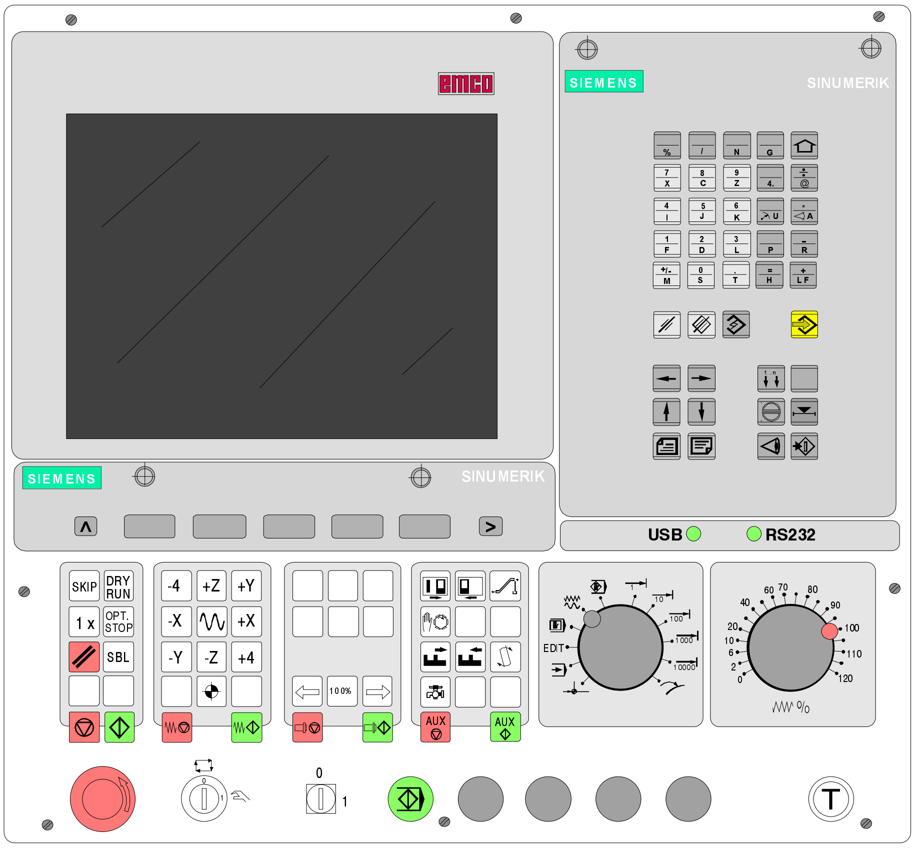
H: Alarms and Messages

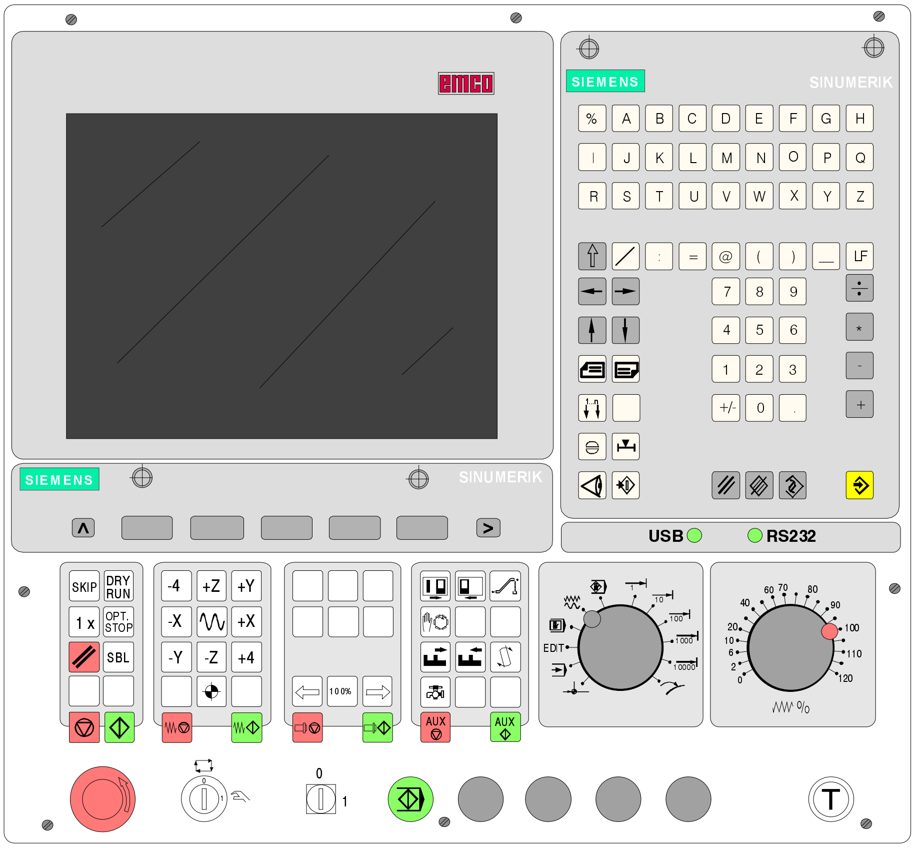
Starting Information

see attachment

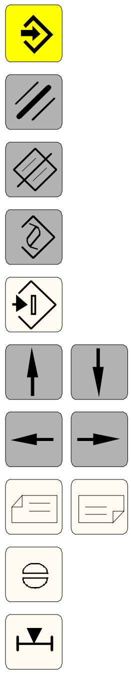
A: Key Description

Control Keyboard, Digitizer Overlay





Key Functions



Input key

Delete input / operator message

Delete word / block

Alter word

Search address / block / word

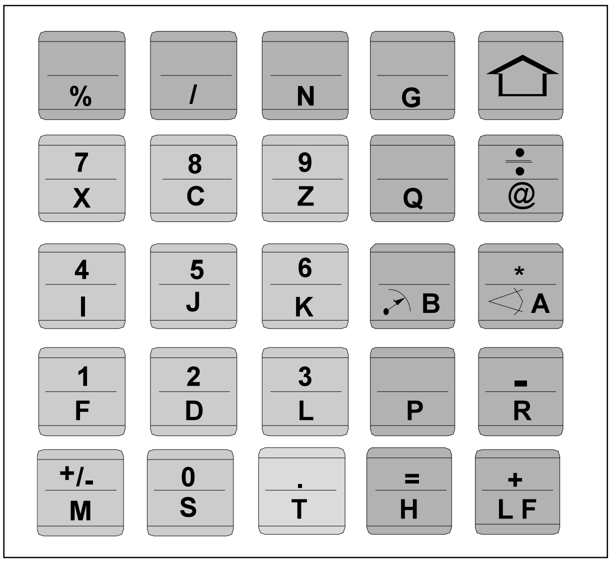
Cursor up / down

Cursor left / right

Page up / down

Acknowledge alarm

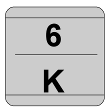
Actual position in double size letters



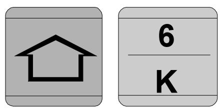
Address and numeric keyboard of the SINUMERIK 810 M

Address and Numeric Keyboard

SINUMERIK 810M:
With the SHIFT key (at the top right edge) you can select the second key function.
Pressing again this key selects the first function again.
After input of a NC address (letter) the SHIFT functions is active automatically



function K

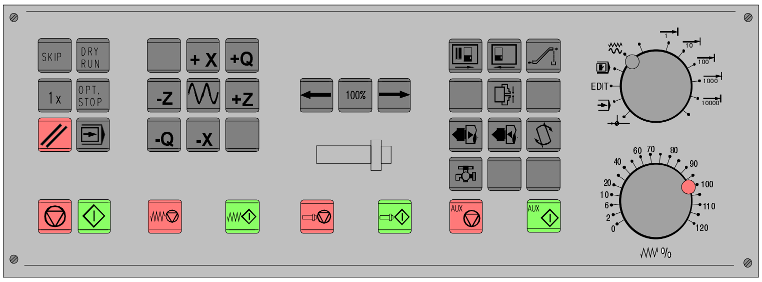


function 6

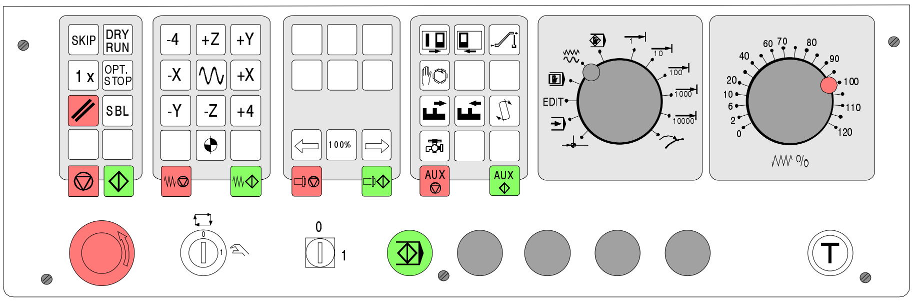
SINUMERIK 820M:
Every address or number has its own key.

Machine Control Keys

The machine keys are in the lower part of the control keyboard or digitizer overlay.
Depending on the used machine and accessory not all of these functions are active.



Machine control keyboard



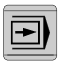
Machine control keyboard of the EMCO PC- Mill Serie

SKIP (skip blocks will not be executed)

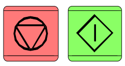
DRY RUN (test run of programs)

OPT STOP (program stop at M01)

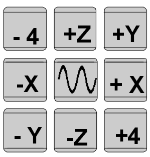
RESET



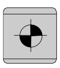
Single block machining



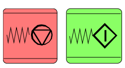
Program stop / program start



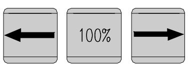
manual axis movement



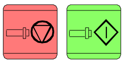
Approaching the reference point in all axes



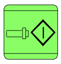
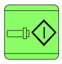
Feed stop / feed start

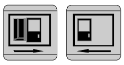


Spindle override lower / 100% / higher



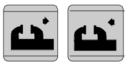
Spindelstop / spindle start; spindle start in JOG and INC1...INC10000 mode:

Clockwise: perss  key short, Counterclockwise: press  min. 1 sec.

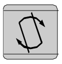


Open / close door

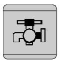
Swivel dividing head



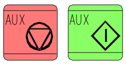
Open / close clamping device



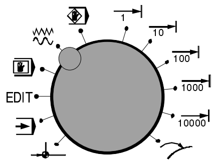
Swivel tool turret



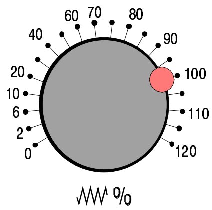
Coolant on/off



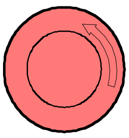
AUX OFF / AUX ON (auxiliary drives off / on)



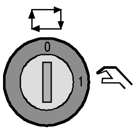
Mode selector



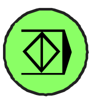
Feed / rapid feed override switch



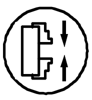
EMERGENCY OFF (Unlock: pull out button)



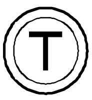
Keyswitch for special operations (siehe Maschinenbeschreibung)



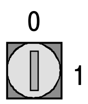
Additional NC start key



Additional key clamping device

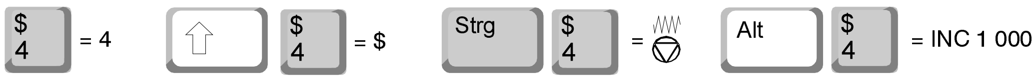


Consent key



No function

PC Keyboard



Some alarms will be acknowledged with the key ESC.

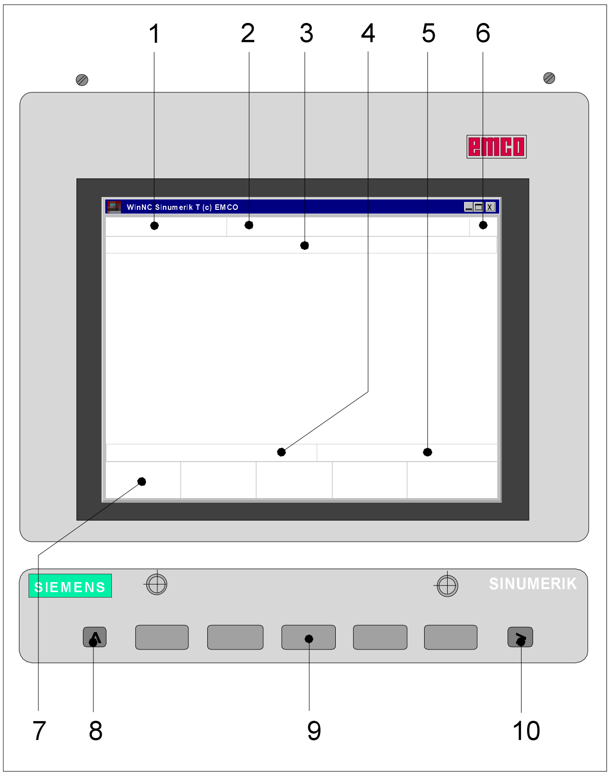
By pressing the key F1 the modes (JOG, AUTOMATIC,...) will be displayed in the softkey line.
The assignement of the accessory functions is described int the chapter "Accessory Functions".

The meaning of the key combination ctrl 2 depends on the machine:
EMCO PC MILL 50/55: Puff blowing ON/OFF
EMCO PC MILL 100/125/155: coolant ON/OFF

* With F12 the function keys PRESET, MDI-AUTOM., JOG, REPOS, AUTOMATIC and REF POINT will be displayed in the softkey line.

The machine functions in the numeric key block are active only with active NUM lock.

Screen with Softkeys



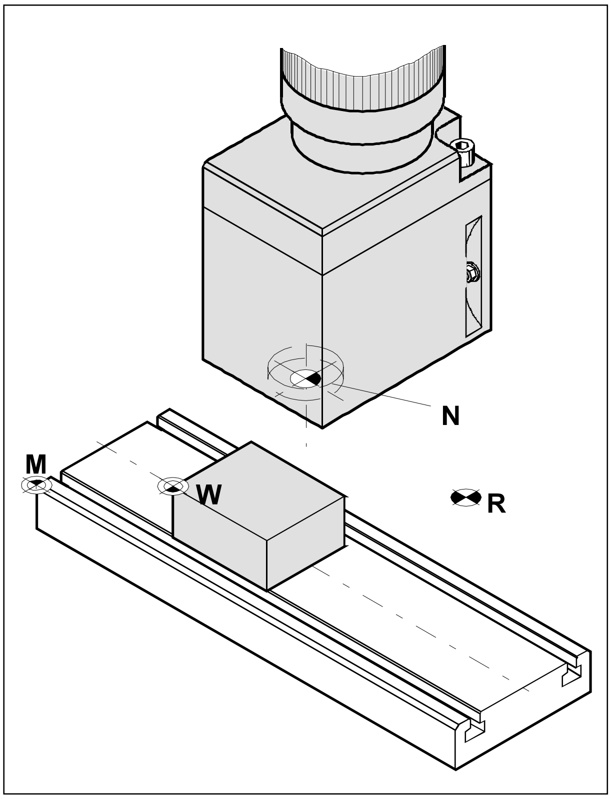
Screen with softkeys

At the operating field the following parts are defined:

1 Display of the mode
2 Display of the operating conditions
3 Display of alarm number, text (comment)
4 Display of notes to the operator
5 Display of inputs from the keyboard
6 Display of the channel number
7 Display of the softkey functions
8 Key "jump back to a higher level menu" (key F2 at the PC)
9 Softkeys (keys F3 - F7 at the PC)
10 Key "Further functions in the same menu" (key F11 at the PC)

Softkeys (9) are keys with multiple meaning. The valid meaning will be displayed at the bottom line (7) of the screen.

B: Basics



Reference points in the working area

Reference Points of the EMCO Milling Machines

M = Machine zero point

An unchangeable reference point established by the machine manufacturer.
Proceeding from this point the entire machine is measured.
At the same time "M" is the origin of the coordinate system.

R = Reference point

A position in the machine working area which is determined exactly by limit switches. The slide positions are reported to the control by the slides approaching the „R“.
Required after every power failure.

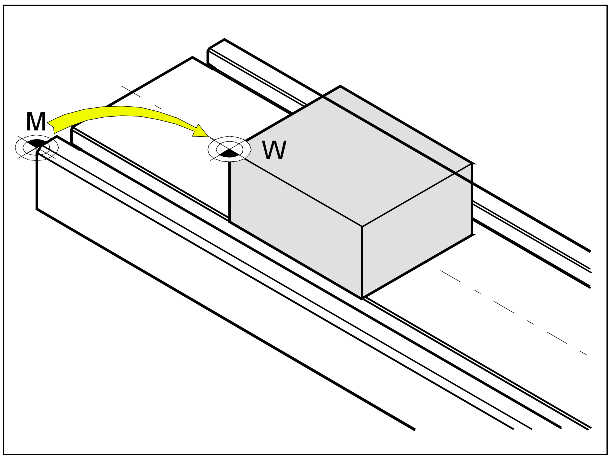
N = Tool mount reference point

Starting point for the measurement of the tools. „N“ lies at a suitable point on the tool holder system and is established by the machine manufacturer.

W = Workpiece zero point

Starting point for the dimensions in the part program.
Can be freely established by the programmer and moved as desired within the part program.

Zero offset



Zero offset from machine zero point M to workpiece zero point W

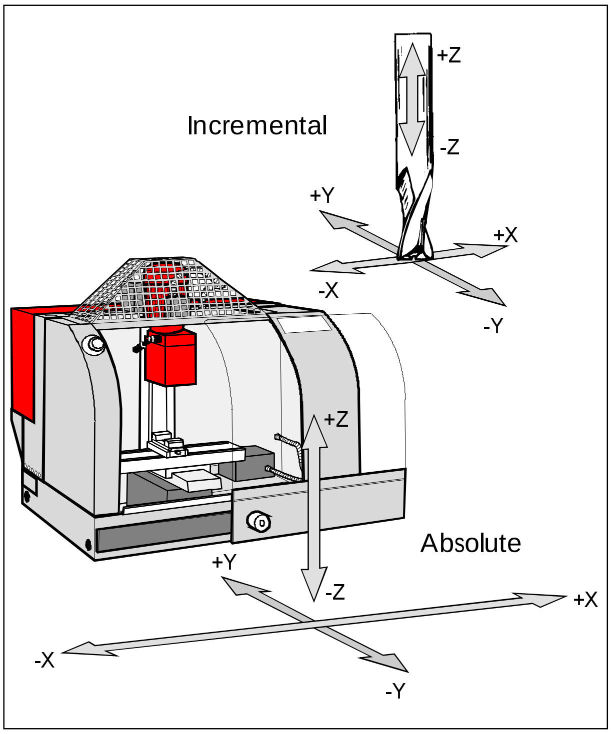
With EMCO milling machines the machine zero point "M" lies on the left front edge of the machine table. This position is unsuitable as a starting point for dimensioning. With the so-called zero offset the coordinate system can be moved to a suitable point in the working area of the machine.

In the setting data zero offset are four adjustable zero offsets available.

When you define a value in the offset register, this value will be considered with call up in program (G54 - G57) and the coordinate zero point will be shifted from the machine zero M to the workpiece zero W.

The workpiece zero point can be shifted within a program with "**G58, G59 - programmable zero offset**" in any number.
More informations see in command description G58, G59.

Coordinate System



Absolute coordinates refer to a fixed point, incremental coordinates to the tool position

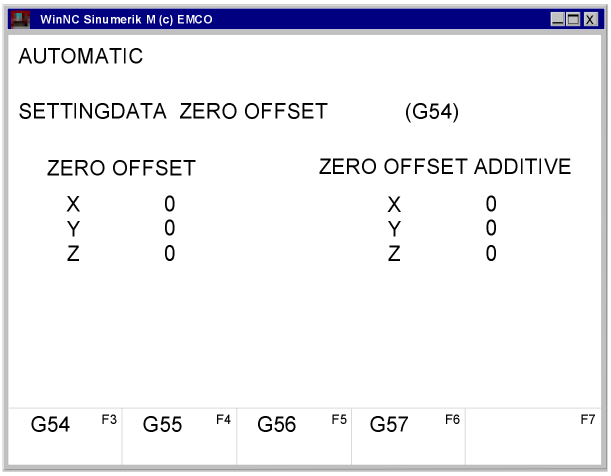
The X coordinate lies parallel to the front edge of the machine table, the Y coordinate lies parallel to the side edge of the machine table, the Z coordinate is vertical to the machine table.
Z coordinate values in minus direction describe movements of the toolsystem towards the workpiece, values in plus direction away from the work piece.

Coordinate System with Absolute Programming

The origin of the coordinate system lies in the machine zero point "M" or after a zero offset in the work piece zero point "W".
All target points are described from the origin of the coordinate system by indication of the respective X, Y and Z distances.

Coordinate System with Incremental Programming

The origin of the coordinate system lies at the tool mount reference point "N" or at the tool tip after a tool call-up.
With incremental programming the actual pathes of the tool (from point to point) are described.

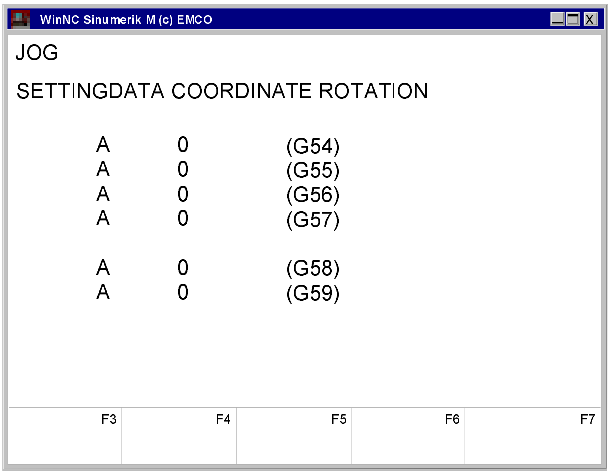


Input pattern for zero offset G54

Input of the Zero Offset

Four zero offsets can be entered (e.g. for four different clamping devices).

- Press the softkey SETTING DATA in any mode.
- Press the softkey ZERO OFFSET.
- The screen shows the input pattern for zero offset G54. The particular offsets G54 - G57 can be selected with softkeys.
- Below ZERO OFFSET the measured values (e.g.: X, Y, Z = distance machine zero point - workpiece zero point) are entered.
- Corrections to this values can be entered below ZO ADDIT.. These corrections will be added.
- Move the Cursor to the value to be altered with the keys .
- Enter new value and press the key .
- The inverse input mark jumps to the next input field.



Input pattern for coordinate rotation

Input of the Coordinate Rotation

A coordinate rotation can be programmed for every zero offset. This coordinate rotation becomes active with calling up the zero offset.

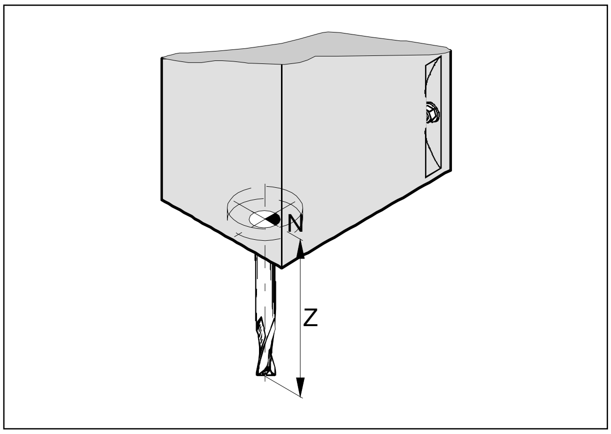
- Press the softkey SETTING DATA in any mode.
- Extend the softkey line (key >) and press the softkey ROTAT. ANGLE.
- The screen shows the input pattern for the coordinate rotation. The single rotations for G54 - G57 can be entered in this pattern, the rotations for G58 and G59 will be determined in the CNC program.
- Move the cursor to the value to be altered with the keys .
- Enter the new value and press the key .
- The inverse input mark jumps to the next input field.

Tool Data Measuring

Aim of the tool data measuring:
The control should use the tool tip or the tool centre point for positioning, not the tool mount reference point.

Every tool which is used for machining has to be measured. The distance between tool tip and tool mount reference point is to be measured.

In the so-called tool register the measured length corrections, the cutter radius and the cutter position can be stored.
Every tool offset number D1 - D99 is related to a tool.
The correction number can be any register number, but has to be considered with tool call in program.

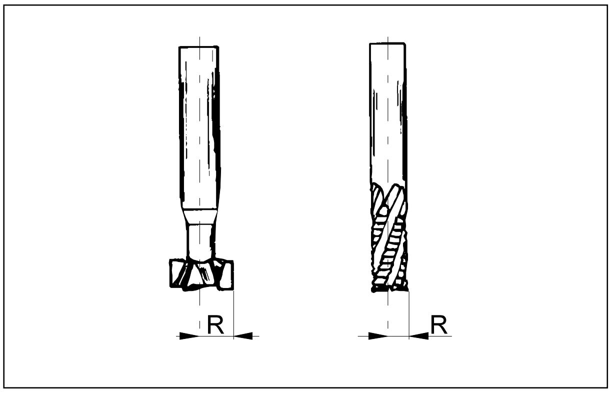


Length correction

Example
The length corrections of a tool in te tool turret station 4 have been stored as correction number 41.

Tool call in program: **T4 D41 L96**
or: **T4 D41 M6**

The address T marks the position in the tool turret, the address D marks the correction number belonging to it. The cycle L96 includes the execution of the tool change (depending on the machine) in the program. Use the command M6, if you want to change the tool with OVERSTORE.



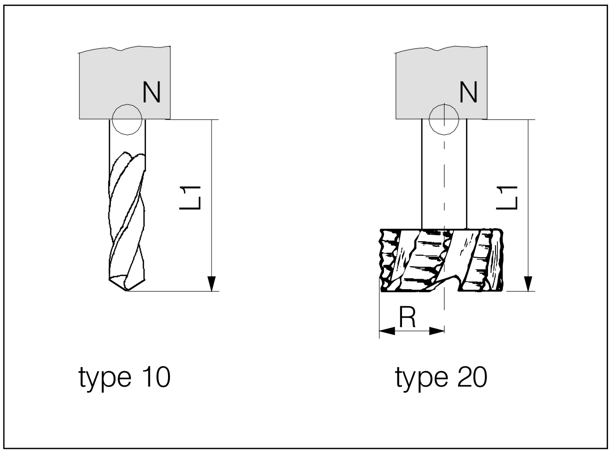
Cutter radius R

Inserting cutter radius is only necessary for using **cutter radius compensation** with this tool.

For active X-Y plane (G17):
The tool data measuring occurs for:
L1: in Z direction absolute from point "N"
R: cutter radius
Type: cutter position 1 - 9

The tool data measuring occurs for type 10:
L1: in Z direction absolute from point "N"
Type: drilling tool 10, milling tool 20

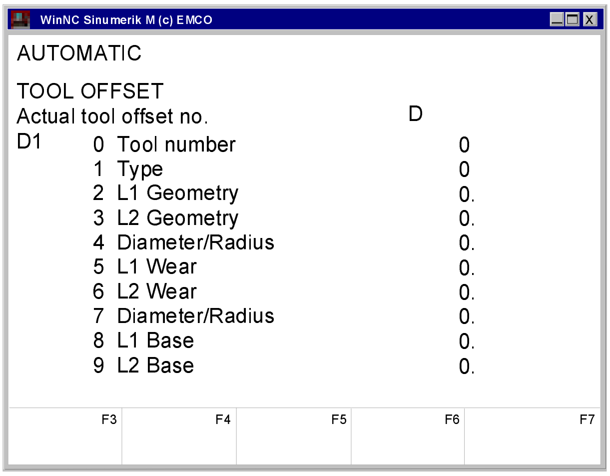
For all other active planes L1 is always calculated as vertical axis to the active plane.
In the following the common case G17 is described.



Tool type

With "**WEAR**" occurs the correction of not exact measured tool data or of worn tools after several machining runs. The inserted length will be added or subtracted from the geometry of the tool incrementally.

L1 +/- ..... Incremental
R +/- ...... Incremental



Input pattern for tool data

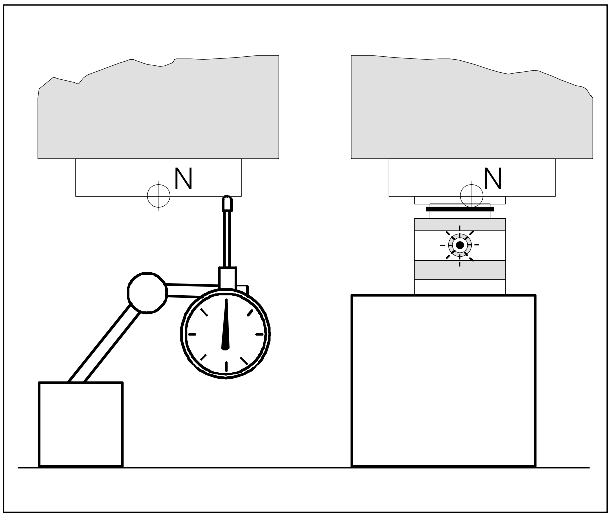
Input of Tool Data

Select the softkey TOOL OFFSET in any mode.
The screen shows the input pattern for tool data.

- Select the desired tool offset number with the keys and or by entry of the correction number and the key "search" (e.g. 5).
- Position the Cursor (invers mark) with the keys and to the desired input field.
  Enter the desired value with the numeric keyboard.
  The entered value will be shown at the input line of the screen.
- Store the correction value with the key in the tool offset register.
  The cursor jumps to the next input position resp. after input of the last value to the first value of the next tool offset number.

  Additive input with , delete with .

Tool Data Measuring with a Metering Clockwork or a Measuring Cell

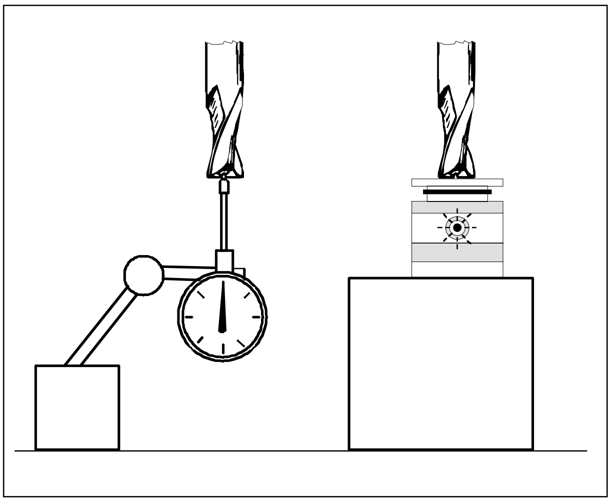


Traverse with the tool mount reference point onto the metering clockwork or measuring cell

Sequence of operation

- Set up the metering clockwork or the measuring cell so in the working area, that the measuring point can be reached with the tool mount reference point and with all tools to be measured.
- Change to the mode JOG.
- Traverse with the tool mount reference point onto the metering clockwork and set it to zero or onto the measuring cell until the lamp lights up.
- Note the first Z value (Z1), which is displayed at the screen.

Note
At the EMCO PC MILL 100 the tool mount reference point is on the face centre of the reference tool. Clamp this tool for the sequence described above.



Traverse with the tool tip onto the metering clockwork or measuring cell

- Clamp the tool to be measured and traverse with it onto the metering clockwork until 0 is displayed or onto the measuring cell until the lamp lights up.
- Note the second Z value (Z2), which is displayed at the screen.
- The difference Z2 - Z1 is the length correction L1 of the tool.
- Insert L1 for the corresponding tool correction number into the input pattern for tool data.
- Insert tool type 10 or 20, for type 20 also the tool radius.

- Clamp next tool and traverse onto the metering clockwork or measuring cell etc..

C: Operating Sequences

Survey Modes

AUTOMATIC

For working off a part program the control calls up block after block and interprets them.
The interpretation considers all corrections which are called up by the program.
The so-handled blocks will be worked off one by one.

JOG

With the JOG keys the tool can be traversed manually. In the submode OVERSTORE (softkey) you can switch on the spindle and swivel the tool holder.

MDI-AUTOMATIC

You can enter blocks of a part program in the intermediate store.
The control works off the inserted block and deletes the intermediate store for new entries.

REFPOINT

This mode is used to approach the reference point. With reaching the reference point the actual position store is set to the value of the reference point coordinates. By that the control acknowledges the position of the tool in the working area.
With the following situations the reference point has to be approached:

- After switching on the machine
- After mains interruption
- After alarm "Approach reference point" or "Ref.-point not reached".
- After collisions or if the slides stucked because of overload.

INC FEED 1 ... INC FEED 10 000

1 ... 10000

In this mode the slides can be traversed for the desired increment (1 ... 10000 in µm / 10^{-4} inch) with means of the JOG keys **-X** **+ X** **- Y** **+Y** **-Z** **+Z**.

The selected increment must be larger than the machine resolution (smallest possible traverse distance), otherwise no movement occurs.

Approach the Reference Point

By approaching the reference point the control will be synchronized with the machine.

- Select the mode REFPOINT (after starting the software REFPOINT is active automatically).

  Press the JOG keys -X or + X to approach the reference point in X, analogous for Y and Z.
- With the key REF ALL all axes will be approached automatically (PC keyboard).

Danger of collisions

Take care of obstacles in the working area (clamping devices, clamped workpieces, etc.).

After reaching the reference point the position of the reference point will be displayed at the screen as actual position. Now the control is synchronized with the machine.

Input of the Gear Position

(only for EMCO PC Turn 50)
For that the control can supervise the correct spindle speed, the selected gear (belt) position of the machine must be entered.

- Press the softkey SETTING DATA in any mode.
- Extend the displayed softkey menu with

  the key >.
- Press the softkey spindle.
- Move the cursor to the input field (Spindle gear stage" and enter the corresponding gear position..

  1 gear position 1 120 - 2000 U/rev
  2 gear position 2 280 - 4000 U/rev

Setting of Language and Workpiece Directory

- Press softkey SETTING DATA
- Expand the softkey line with the key > and press the Softkey GENERAL DATA
- In the input pattern you can set the language and the work piece directory.

Workpiece directory

In the workpiece directory the CNC programs created by the operator will be stored.
The workpiece directory is a subdirectory of the directory in which the software was installed.
Enter the name of the workpiece directory with the PC keyboard, max. 8 characters, no drives or pathes. Not existing directories will be created.

Active language

Selection from installed languages, the selected language will be activates with restart of the software.
Input with PC keyboard:

- DT for German
- EN for English
- FR for French
- SP for Spanish
- NL for Netherlands

Input of Programs

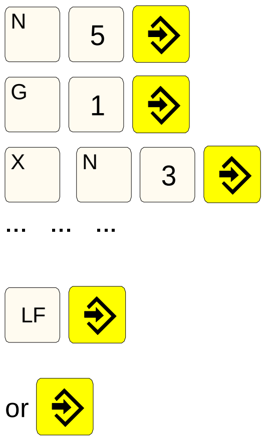
Part programs and subroutines can be entered in the modes
JOG
AUTOMATIC
INC 1 ... INC 10 000 and
REFPOINT.

Call up an existing or new program

- Press softkey PART PROGRAM
- Press softkey EDIT
- Enter program number %... or L...
- Press softkey SELECT PROGRAM
  Blocks in an existing program will be displayed.

Input of a Block

Example:



Block number (not necessary)

1. word

2. word

LineFeed - block end (with PC keyboard)

Insert Block

Position the cursor before the block, that should follow the inserted block and enter the block to be inserted.

Delete Block

Position the cursor before the block, enter block number (if no block number: N0) and press key .

Insert Word

Position the cursor before the word, that should follow the inserted word and enter the word (address and value) to be inserted and press .

Alter Word

Position the cursor before the word to be altered, enter word and press .

Delete Word

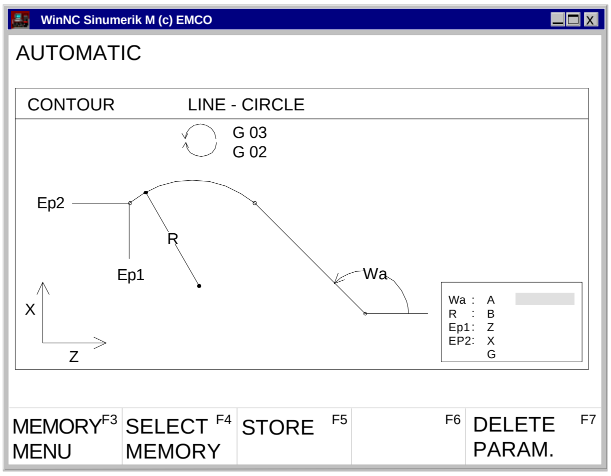
Position Cursor before the word to be deleted, enter address (e.g. X) and press key .

Program Input with Guiding Function

In an openend program you can enter blocks guided by menues.
Frequently used G and M commands are displayed as softkey functions.
It is also possible to enter predefined contour drafts without need to calculate intermediate positions.

Example: Program contour draft line - circle

- Press softkey GUIDING
- Press softkey GEOM. PATH
- Press softkey LINE - CIRCLE
- The input pattern beside will be displayed. The selected contour draft (line-circle) is displayed graphically.
- Enter with the keyboard the input values one by one.
- If in the input fields several values are in curved brackets only one of this values must be entered. If you have entered several values, you can delete the odd values with the softkey PARAM. DELETE.
- Press the softkey STORE, SOTRE MENU or STORE SELECT when input is finished.
- Now the contour draft will be stored as block in the program with all the entered geometry values. The software automatically creates a block end (LF) and displays the inserted block.



Input pattern for guiding line-circle

Program Input with CAD/CAM Systems

Principially NC programs can be read in from CAD/ CAM systems into WinNC SINUMERIK 810/820 M.

Act as following:

- The NC program has to be created in the format of the SINUMERIK 810/820.
- The file has to be renamed.
  The NC programs of the Control System are stored as following:
  %MPFxxxx ...... main program
  %SPFxxxx ...... subprogram
  (xxxx .... program number)
  E.g.: Rename with WINDOWS File Manager:
  From: PART1.81M
  To: %MPF123
- Import the program with DATA IMPORT (see Data Input/Output).

Program Administration

- Press softkey PART PROGRAM
- Press softkey PROGR.- HANDLE
- In the softkey line the functions
  COPY
  RENAME
  DELETE
  will be displayed

Copy Program

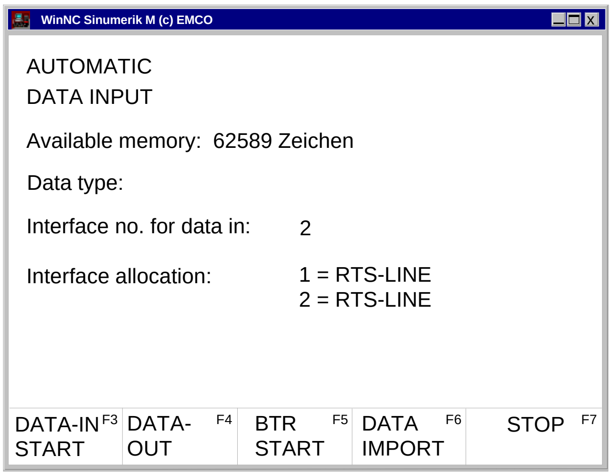
Example:
- Enter with keyboard
  %88=%5
- Press the softkey COPY
- The software copies the program %88 and stores it again with the program number %5.
  The program %88 is still existing.

Rename Program

Example:
- Enter with keyboard
  %12=%15
- Press the softkey RENAME
- The software renames the program %12 to %15.
  The program %12 will not be kept.

Delete Program

Example:
- Enter with keyboard
  %22
- Press the softkey DELETE
- The software deletes the program %22.



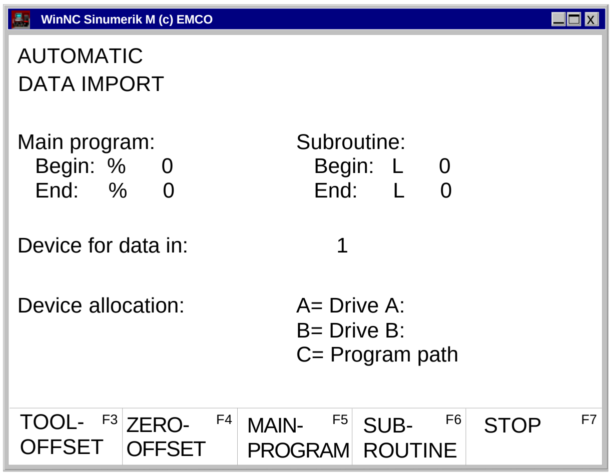
Input pattern for data input-output

Data Input-Output

- Press softkey DATA IN-OUT
- The input pattern beside will be displayed
- With "Interface no. for data in:" you can select a serial interface (1 or 2) or a drive (A, B or C).
  1 serial interface COM1
  2 serial interface COM2
  A disk drive A
  B disk drive B
  C hard disk drive C, workpiece directory (can be determined with installation or with GENERAL DATA in SETTING DATA) or import/export directory (see WinConfig, 4.1 Alter Directories)

Data Input via COM1 / COM2

- Press softkey DATA IN START. This will start the receiving function of the software.
- At the right top edge of the screen DIO (Data Input/Output) will be displayed. Destination signs the data from the sender (punched tape drive, ...).
- Start the sender.
- With the softkey STOP you can abort the data input at any time, with DATA IN START you can restart data input.
- A direct call-up of certain data by the software is not possible with data input.

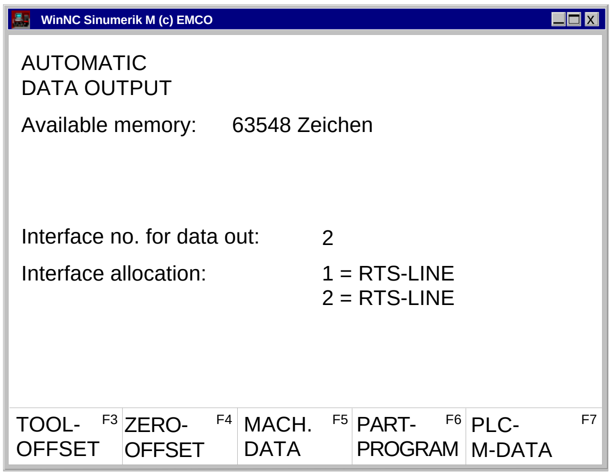


Input pattern for data import

Data Import

With the function data import you can load data from the drives A, B and C.

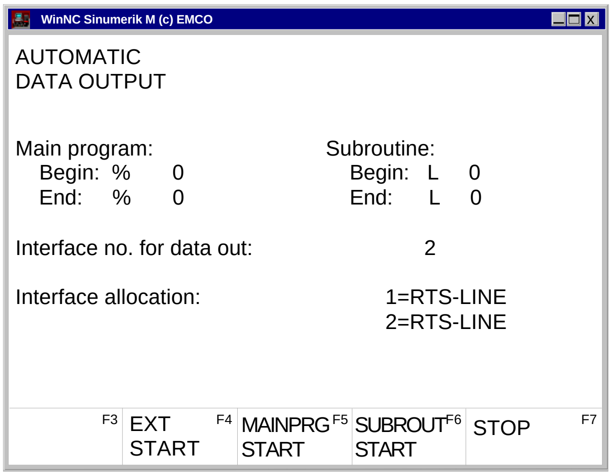
- Press softkey DATA IMPORT
- Select drive (device for data in)
- Enter below "Mainprogram" resp "Subprogram" the following program numbers:
  Begin: the first program to read in
  End: the last program to read in
- Press the softkey MAIN PROGRAM or SUB-ROUTINE to start reading in the data.
- Transmission of zero offsets, tool data:
  Press the softkey START.
- With the softkey STOP you can interrupt the data input at any time.



Input pattern for data output

Data Output

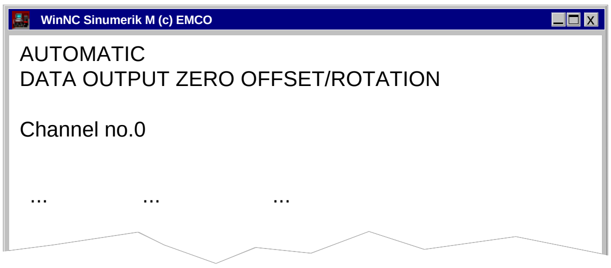
- Press softkey DATA OUT.
- The screen shows the input pattern beside.
- With "Interface no. for data out" you can enter a serial interface (1 or 2) or a drive (A, B or C).
- If you send data to disks, this data will be sent in the same format as with output to the serial interface. This data have to be read in with DATA IMPORT and must not be copied directly into the workpiece directory.



Input pattern for data output - part program

Example: Program output

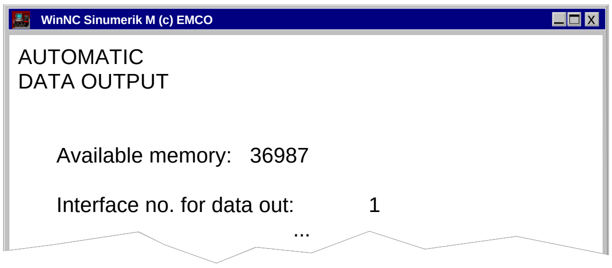
- Press softkey PART PROGRAM
- The screen shows the input pattern beside
- Enter below mainprogram resp. subprogram the following numbers:
  Begin: the first program to be sent
  End: the last program to be sent
- Press softkey MAINPRG START resp. SUBRROUT START. This will start the send function of the software.
- With the softkey STOP you can interrupt data output at any time, with MAINPRG START resp. SUBROUT START you restart data output with the first program to be sent.



Data output zero offset - rotation

Example: Zero offset output

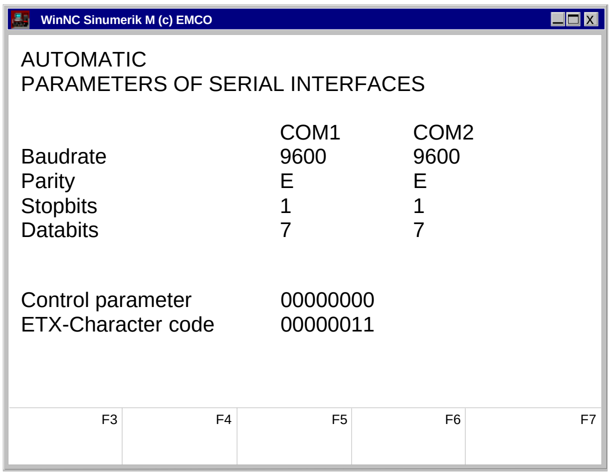
You can select to put out zero offsets or coordinate rotations.
Channel-no. 0: Output zero offsets
Channel-no. 1: Output coordinate rotation



Input pattern for data output - printer

Print Data

- Press softkey DATA OUT
- The screen shows the input pattern beside.
- With "Interface no. for data out" you can enter P to select a printer.
- Act like data output



Input pattern for adjusting the serial interface

Adjusting the Serial Interface

- For transmission the interfaces of sender and receiver have to be set similar.
- Select SETTING DATA - SETTING BITS with softkeys.

Settings:

Baudrate	110, 300, 600, 1200, 2400, 4800, 9600, 19200
Parity	E, O, N
Stop bits	1, 2
Data bits	7, 8

Data transmission from / to original control in ISO-Code only.
ISO: 7 Data bits, Parity even (=E)

Control parameter:
Bit 0: 1...Transmission will be ceased only with ETX-(End of Text) Code (not with M30, M17, M2) - this enables transmitting several programs in one sequence
Bit 7: 1...Overwrite part program without message
0...Message, if a program is already existing

ETX character code:
free setable, has to be conform with the code set at the CNC

Settings at the original control 810/820 with SETTING DATA - SETTING BITS:

5010 - 5013:	first interface
5018 - 5021:	second interface
5028:	ETX sign

5010, 5012: 00000000
5011, 5013: 11000xxx
coded baud rate:

001:	150 bd	010:	300 bd	011:	600 bd
100:	1200 bd	101:	2400 bd	110:	4800 bd
111:	9600 bd				

NOTE

When you use an interface expansion card (e.g. for COM 3 and COM 4), take care that for every interface a separate interrupt is used (e.g.: COM1 - IRQ4, COM2 - IRQ3, COM3 - IRQ11, COM4 - IRQ10).

Program Run

Start of a Part Program

Before starting a program the control and the machine must be ready for running the program.
- Select the mode AUTOMATIC.
- Enter the desired part program number (e.g.: %59: % 7 9 .
- Press the key .

Messages while Program Run

In the first line on the screen the influences on program run will be displayed.

HALT: AUTO interrupted

The mode was changed or the key was pressed.

HALT: Single block
In single block mode one block was worked off (finished). Go on in program with key .

HALT: Pr. stop M00, M01
Programmed interruption of the program process. Continue the program with key .

HALT: Read enable
Read enable is a PLC output signal. The current block is not finished (e.g. with tool change). The next program block will be worked off after finishing the current block.

HALT: Dwell time
Processing the program is interrupted for the duration of the programmed dwell time.

FST
FEED STOP. This message will be displayed, if the PLC stops the program to execute an operation (e.g. tool change).

Program Influence

By actuating the following softkeys runnig programs can be influenced.

- Press the softkey PROGRAM CONTROL in the mode AUTOMATIC or MDI-AUTOMATIC.
- The menu line shows the following softkeys:

SKIP YES-NO	(skip block)
DRY ON-OFF	(dryrun)
OPT.STOP YES-NO	(programmed stop)
DEC-SBL YES-NO	(decoding single block)

Select this functions by pressing the corresponding softkey, deselect by pressing the softkey again.

Skip block:
Blocks in the program, which are marked by a slash before the block number (/N ...), now will not be worked off while program run.

Dryrun:
For test run without workpiece this function can be activated. All blocks with programmed feed (G01, G02, G03, G33, ...) traverse with dryrun-feed instead of the programmed feed.
The dryrun feed can be set in the setting data.

Programmed stop:
When an M01 is present in the part program, the program normally is not stopped. When the softkey function is marked with YES, the program stops with M01.

Decoding single block:
This function works in a similar way like the function SINGLE BLOCK. If this function is activated by YES, after every block, which is running through decoding, the part program will be stopped.

With the key the program can be continued.

As difference to the normal single block mode the decoding single block mode stops also with calculation blocks.
Application: testing cycles.

Overstore

The function OVERSTORE changes one or several values in the intermediate store.
For OVERSTORE the program has to be stopped.

- Press key
- Extend softkey line (key >) and press softkey OVERSTORE.
- Now you can enter new values for tool position T, spindle speed S, auxiliary function H and miscellaneous function M.
- Press the key , to activate the alterations and continue the program.
- The program runs with these new values, until in the program or by repeated OVERSTORE this values will be altered.

Block Search

With this function you can start a program at any block.
While block search the same calculations will be proceeded as with normal program run but the slides does not move.

- Press the key RESET ().
- Select the mode AUTOMATIC.
- Extend displayed softkey line (>) and press softkey BLOCK SEARCH.
- Enter % for a main program resp. L for a subroutine and press the key .
- Enter the program number via keyboard and press the key .
- Enter the block number for the search target via keyboard and press the key .
- Press the softkey START.
  The program blocks will be worked off until the given search target is reached.
- With the key you can activate the program run.

Program Interruption

1. Way:
Change to the modes JOG or INC 1 ... INC 10 000.
With these changes no RESET occurs.
The drives will be stopped with holding the programmed tool path.
Display: HALT: AUTO interrupted

2. Way:

Press the key .

The drives will be stopped with holding the programmed tool path.
Display: HALT: AUTO interrupted

Status Display of the PLC

Only on machines with PLC (**P**rogrammable **L**ogical **C**ontrol) - e.g. PC TURN 50 with tool turret.

- Press softkey DIAGNOSIS
- Press softkey PLC- STATUS

The statii to be displays can be entered directly
e.g.: EB10,H

H hex
B binary
D decimal

The data format (H, B, D) need not to be determined. If no format is determined the format is like shown in the table.

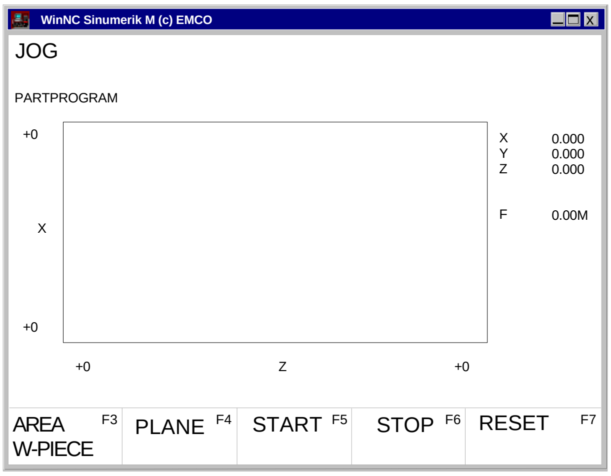
Name	Address area	Remark	Format
E	0 - 127	input byte	B
A	0 - 127	output byte	B
M	0 - 255	marker	B
S	0 - 255	S-marker	B
DB	0 - 255	data module	D
DW	0 - 255	data word*	H
Z	0 - 31	counter	D
T	0 - 63	time	D

*... before DW can be displayed, with DB a data module has to be selected.

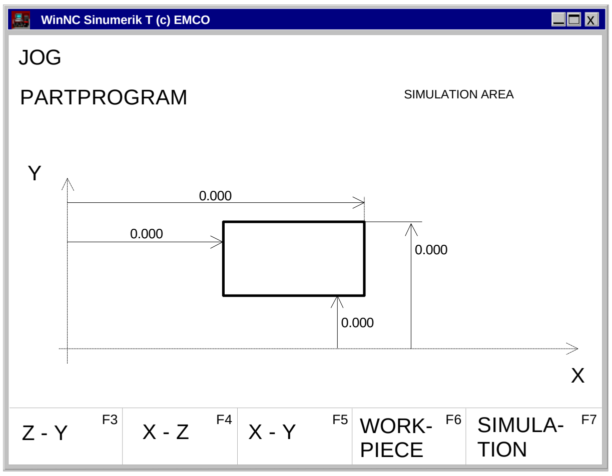
Display of the Software Versions

- Press softkey DIAGNOSIS
- Press softkey SW VERSION

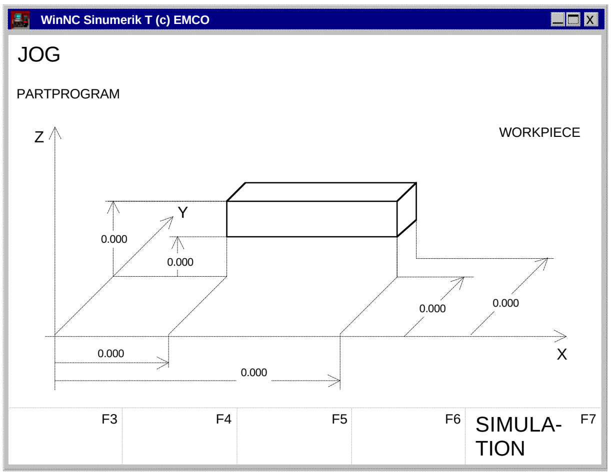
The software version of WinNC and the eventually connected RS 485 devices will be displayed.



Graphic screen



Define workpiece size and simulation area



Define work piece size

Graphic simulation

NC programs can be simulated graphically.

- Press the softkeys PART PROGRAM and EDIT.
- When no program is selected, enter the program number and press the softkey SELECT PROGRAM.
- The softkey 3DVIEW starts the 3D graphic simulation Win 3D View. This is an option and not included in the basic version of WinNC.
- Start the graphic simulation with the softkey SIMULATION.
- The screen shows the graphic simulation.
- Press the softkey AREA, to define the workpiece size and the simulation area.
- The bold lined rectangular shows the visible screen window for simulation.
  The dialogue line shows the meaning of the marked dimension.
- Enter the measures of the screen window.
- Select the plane for simulation with the softkeys Z-Y, X-Z and X-Y.
- Select the softkey WORKPIECE to determine the size of the workpiece.
- The workpiece is show three-dimensionally.
  The dialogue line shows the meaning of the marked dimension.
- Enter the measures of the workpiece.
- With START the graphic simulation starts.
- With STOP the graphic simulation stops.
- With RESET the graphic simulation will be aborted.

Movements in rapid traverse will be displayed as dashed lines, movements in working traverse will be displayed as full lines.

D: Programming

Program Structure

NC programming for machine tools according to DIN 66025 is used.
The NC program is a sequence of program blocks which are stored in the control.
With machining of workpieces these blocks will be read and checked by the computer in the programmed order.
The corresponding control signals will be sent to the machine.

The NC program consists of:

- Program number
- NC blocks
- words
- addresses
- number combinations (for axis addresses partly with sign)

%1234				
N0100	G01	X25	Y20	Z-17
Z-17				
Z				
-17				

Components of an machining program

Used Addresses

% .... program number 1 bis 9999
L .... subroutine number1 bis 9999
N............ block number 1 bis 9999

G .... pathfunction
M .... miscellaneousfunction

A............ angle
D............ tool offset 1 - 49
F .... feedrate, dwell
I, J, K ..... circle parameter, thread pitch
P .... number of subroutine runs, scale factor
R .... interchange parameter for cycles
S .... spindle speed
T .... tool call (tool turret position)
U .... circle radius, radius (pos sign), chamfer (neg. sign)
X, Y, Z ... position data (X also dwell)
L F .... blockend

Survey of G commands

G00 .... Rapid traverse
G01[1] .... Linear interpolation
G02 .... Circular interpolation clockwise
G03 .... Circular interpolation counterclockwise
G04[2] .... Dwell
G09[2] .... Exact stop blockwise
G10 .... Polar coordinate interpolation, rapid traverse
G11 .... Polar coordinate interpolation, linear interpolation
G12 .... Polar coordinate interpolation, circular interpolation cockwise
G13 .... Polar coordinate interpolation, circular interpolation counterclockwise
G17[1] .... Plane selection X-Y
G18 .... Plane selection Z-X
G19 .... Plane selection Y-Z
G25 .... Minimum working area limitation
G26 .... Maximum working area limitation
G33 .... Thread cutting in single blocks
G40[1] .... Cancel cutter radius compensation
G41 .... Cutter radius compensation left
G42 .... Cutter radius compensation right
G48[2] .... Leave as approached
G50[1] .... Cancel scale modification
G51 .... Scale modification
G53[2] .... Cancel zero offset blockwise
G54[1] .... Zero offset 1
G55 .... Zero offset 2
G56 .... Zero offset 3
G57 .... Zero offset 4
G58[2] .... Programmable zero offset 1
G59[2] .... Programmable zero offset 2
G60 .... Exact stop mode
G62 .... Cancel exact stop mode
G64[1] .... Cancel exact stop mode
G70 .... Measuring in inches
G71 .... Measuring in millimeter
G80[1] .... Delete G81 - G89
G81 .... Call cycle L81
G82 .... Call cycle L82
G83 .... Call cycle L83
G84 .... Call cycle L84
G85 .... Call cycle L85
G86 .... Call cycle L86
G87 .... Call cycle L87
G88 .... Call cycle L88
G89 .... Call cycle L89
G90[1] .... Absolute programming
G91 .... Incremental programming
G92 .... Cylindrical interpolation
G94[1] .... Feed rate in minutes
G95 .... Feed rate in revolutions
G147[2] .... Soft approach to contour with linear
G247[2] .... Soft approach to contour with quarter circle
G347[2] .... Soft approach to cont. with semicircle
G148[2] .... Soft leaving to contour with linear
G248[2] .... Soft leaving to contour with quarter circle
G348[2] .... Soft leaving to contour with semicircle

Survey of M commands

M00 .... Programmed stop, unconditional
M01 .... Programmed stop, conditional
M02 .... Main program end
M03 .... Spindle ON clockwise
M04 .... Spindel ON counterclockwise
M05[1] .... Spindle OFF
M06 .... Tool change
M08 .... Coolant ON
M09[1] .... Coolant OFF
M17 .... Subroutine end
M27 .... Swivel dividing head
M30 .... Main program end
M53 .... No mirror axis X
M54 .... Mirror axis X
M55 .... No mirror axis Y
M56 .... Mirror axis Y
M57 .... No mirror axis Z
M58 .... Mirror axis Z
M71 .... Puff Blowing ON
M72[1] .... Puff Blowing OFF

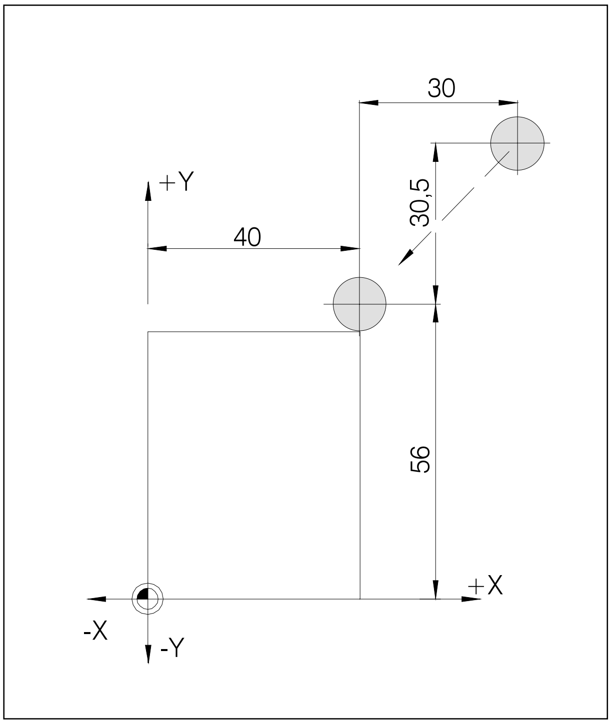
Survey of cycles

L81-89 ... Drilling cycles, comp. G81 - G89
L96 .... Cycle for tool change
L900 ...... Drilling pattern
L901 ...... Milling pattern slot
L902 ...... Milling pattern elongated hole
L903 ...... Mill rectangular pocket
L904 ...... Milling pattern circular slot
L905 ...... Drilling pattern single hole
L906 ...... Drilling pattern row of holes
L930 ...... Mill circular pocket
L999 ...... Clear buffer memory

[1] .... initial status
[2] .... effective blockwise

Command Description G Commands

G00 Rapid Traverse



Absolute and incremental measures

Format
N.... G00 X... Y... Z...

The slides are traversed at maximum speed to the programmed target point (tool change position, start point for following machining).

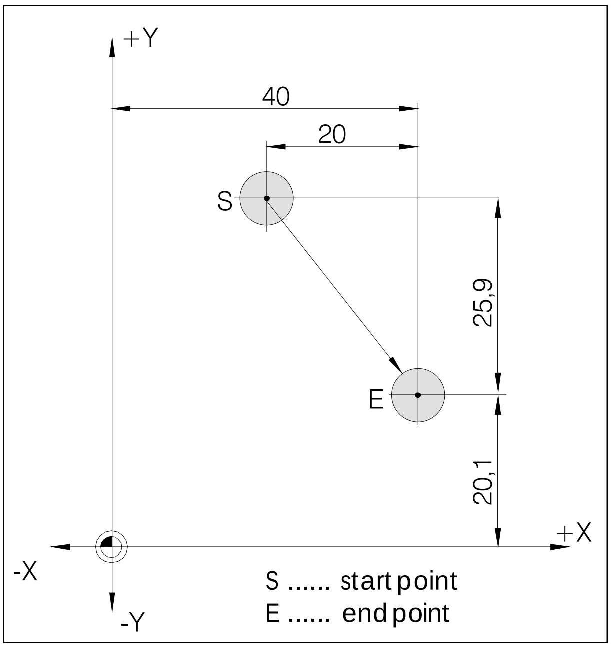
Note
- A programmed feed F is suppressed while G01.
- The maximum feed is defined by the producer of the machine.
- The feed override switch is active.

Example

absolute G90
N50 G00 X40 Y56

incremental G91
N50 G00 X-30 Y-30.5

G01 Linear Interpolation



Absolute and incremental measures

Format
N... G01 X... Y... Z.... F....

Straight movements with programmed feed in mm/rev (initial status)

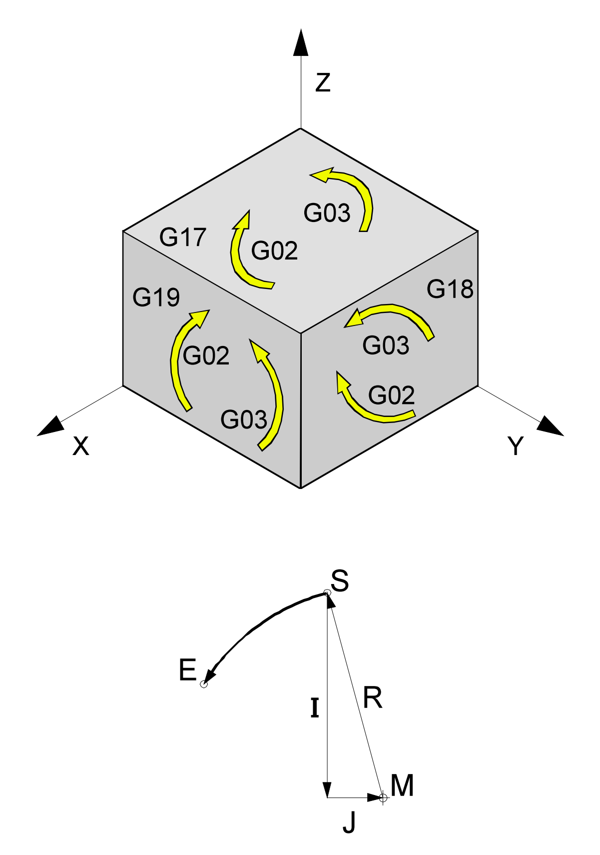
Example

absolute G90
.....
N20 G01 X40 Y20.1 F0.1

incremental G91
.....
N20 G01 X20 Y-25.9 F0.1

G02 Circular Interpolation Clockwise

G03 Circular Interpolation Counterclockwise



Rotational directions of G02 and G03

Format

N... G02/G03 X... Y... Z... I... J... K... F...
or
N... G02/G03 X... Y... Z... U... F...

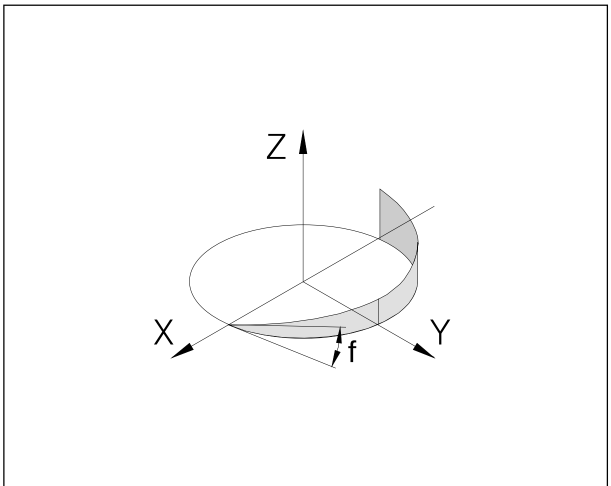
X, Y, Z .... End point of the arc (absolute or incremental)
I, J, K .... Incremental circle parameter (Distance from start point to center of arc, I is related to X, J to Y and K to Z)
U .... Radius of the arc (arc smaller than half circle with +U, larger than half circle with -U) can be entered instead of the parameter I, J, K.

Note

A circular interpolation can be proceeded in the active plane only.

Programming the value 0 for I, J or K can be omitted. The position of the circle end point will be checked, a tolerance of 100µm (computing and rounding errors) is allowed.
The observation of G02, G03 is always vertical to the active plane.

Helix Interpolation



Helix curve

Normally only two axes are determined for an arc. These two axes determine the plane in which the circle lies.
If a third, vertical axis is determined, the movements of the slides are coupled in a manner, that a screw line will be traversed.
The programmed feedrate will not be hold at the real path, but at the circle path (projection). The third, linear linear moved axis will be controlled in a way, that it reaches the end point at the same time like the circular axes.

Limitations

- The helix interpolation is possible only with G17.
- The start angle ϕ has to be less than 45°.
- If the tangents differ more than 2° in the solid angle, an exact stop will be proceeded.

G04 Dwell

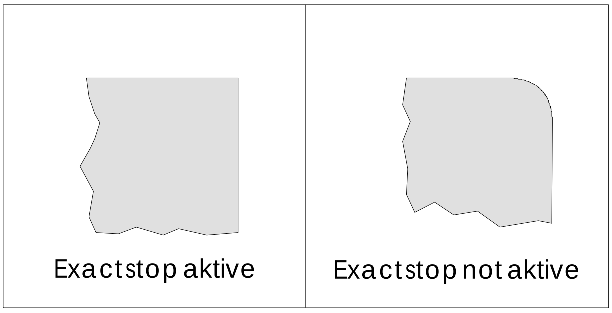
Format
N... G04 X/F [sec]

The tool movements will be stopped for a time defined by X or F (in the last reached position) - sharp edges - transititions, cleaning cut-in ground, exact stop.

Note
The dwell time starts at the moment when the tool movement speed is zero.

Example
N75 G04 X2.5 (dwell time = 2.5 sec)

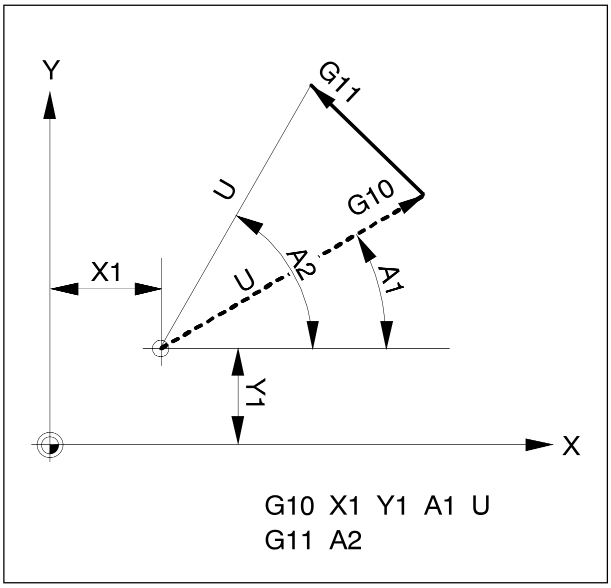
G09 Exact Stop



Format
N... G09

The next block will be worked off after the block with G09 is finished and the slides have reached standstill at the end position.
Edges will not be rounded and precise transititions will be reached.
G09 is effective blockwise.

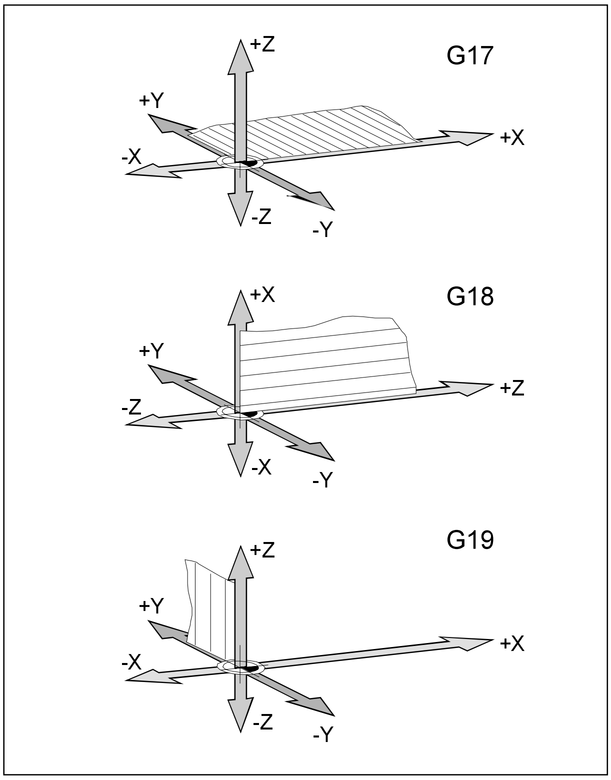
G10 - G13 Polar Coordinate Interpolation



Movements determined by polar coordinates

G10 Rapid
G11 Linear Interpolation
G12 Circular Interpolation clockwise
G13 Circular Interpolation counterclockwise

With angle and radius dimensioned drawings can be entered directly with polar coordinates in the active plane.
To determine the traverse path the control needs the centre point, the radius and the angle.
The centre point will be determined with cartesian coordinates (X, Z) and entered in absolute measure with first programming. A later incremental input (G91) refers always to the last programmed centre point.
The radius will be programmed under address U.
The angle will be programmed under address A.
The angle is 0° in + direction of the axis that was programmed first with centre point. The input of angle is positive (counterclockwise).



Definition of the main planes

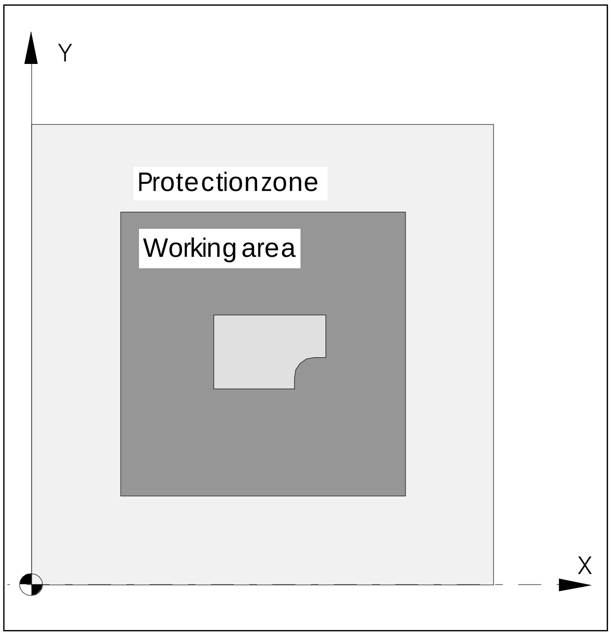
G17-G19 Plane Selection

Format

N... G17/G18/G19

With G17 bis G19 the plane will determined, in which circular interpolation and polar coordinate interpolation can be proceeded and in which the cutter radius compensation will be calculated.
In the vertical axis to the actve plane the tool length correction (tool offset) will be proceeded.

G17 XY plane
G18 ZX plane
G19 YZ plane

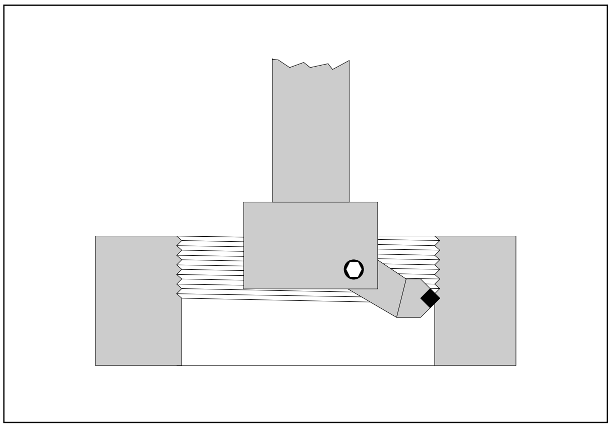


G25/G26 Programmierbare Arbeitsfeldbegrenzung

Format:
N... G25 X... Y...
N... G26 X... Y...

G25 .... minimum working area limitation
G26 .... maximum working area limitation

G25/G26 limitates the working area in which the tool can traverse.
By that in the working area, a safety area can be established which are locked for tool movements.



Application of thread cutting

G33 Thread Cutting

Format

N... G33 X... Z... I/K...

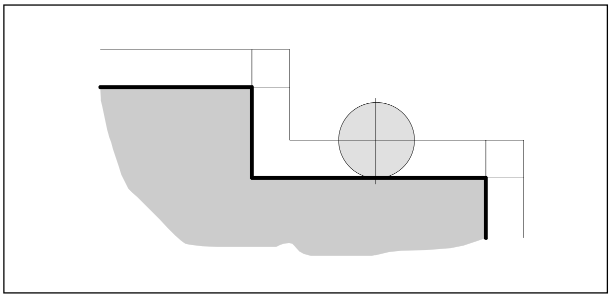
K ....Thread pitch [mm]
Z ....Thread depth [mm]

With a suitable tool threads may be cut.

Note

- Feed and spindle override are not active with G33 (100%)
- G33 works only with the EMCO PC Mill 100, because on the EMCO PC Mill 50 is no milling spindle encoder.

Cutter Radius Compensation

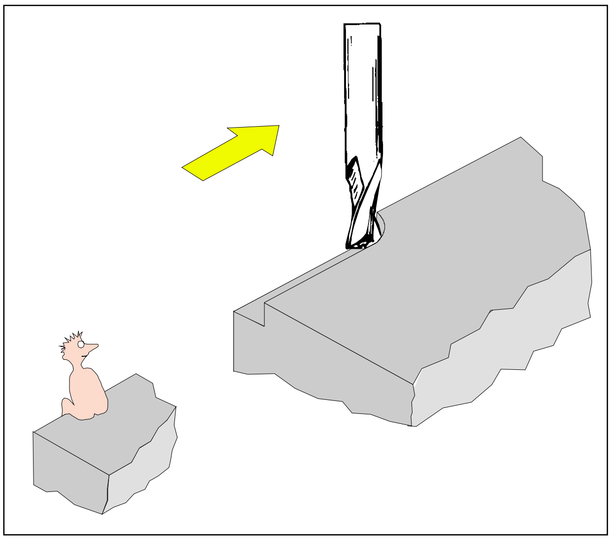


Radius compensated toolpath

With active cutterradius compensation a path parallel to the contour will be calculated automatically by the control and so the radius of the milling cutter compensated.

G40 Cancel Cutter Radius Compensation

The cutterradius compensation will be cancelled by G40.
Cancellation is only allowed with a straight movement (G00, G01).
G40 can be programmed in the same block as G00 resp. G01 or in the block before.
G40 usually will be programmed in the withdrawal block to the tool change point.



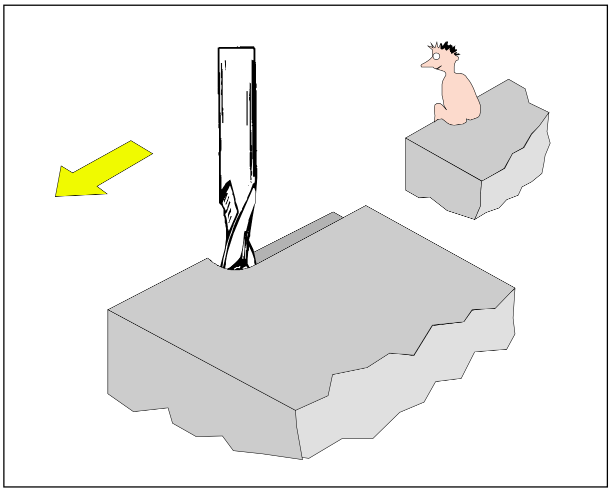
Definition of G41 cutter compensation left

G41 Cutter Radius Compensation Left

If the programmed tool (viewed in the direction of machining) is on the left of the path to be machined, the radius compensation is to be selected with G41. For calculating a radius, with selection of cutter radius compensation a tool offset (D number) has to be active and in the tool register a radius has to be inserted.

Notes

- No direct change between G41 and G42 - cancel with G40 previously.
- Selection is only permitted in conjunction with G00 or G01.
- Entering a cutter radius with tool measuring is necessary.
- Change of tool correction number is not possible with active cutterradius compensation.



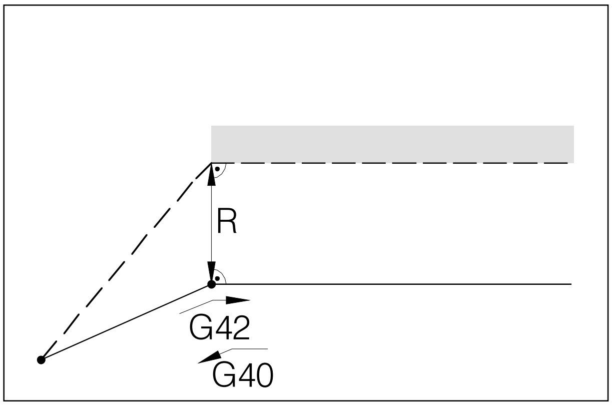
Definition of G42 cutter compensation right

G42 Cutter Radius Compensation Right

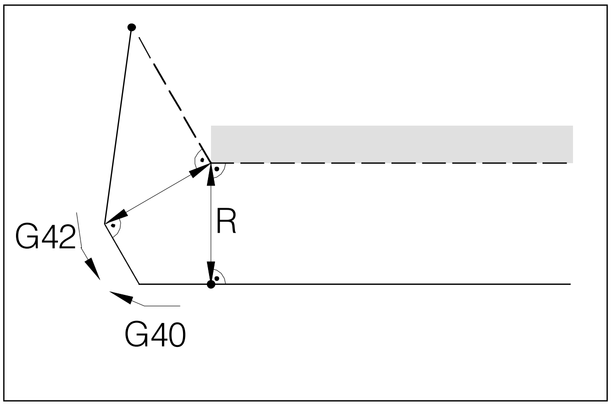
If the programmed tool path (viewed in the direction of machining) is on the right of the material to be machined, the radius compensation is to be selected with G41.

Notes see G41!

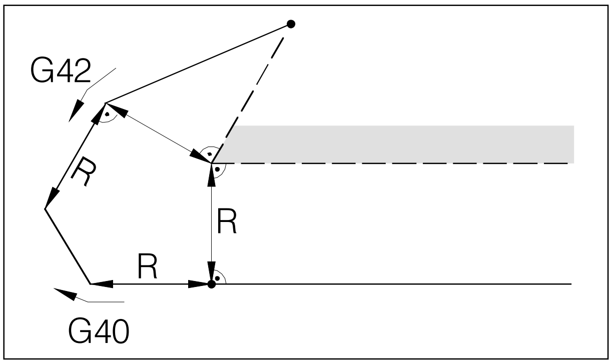
Tool pathes with selection / cancellation of the cutter radius compensation



Frontal approach or leaving of an edge point



Approach or leaving of an edge point side behind



Approach or leaving of an edge point behind

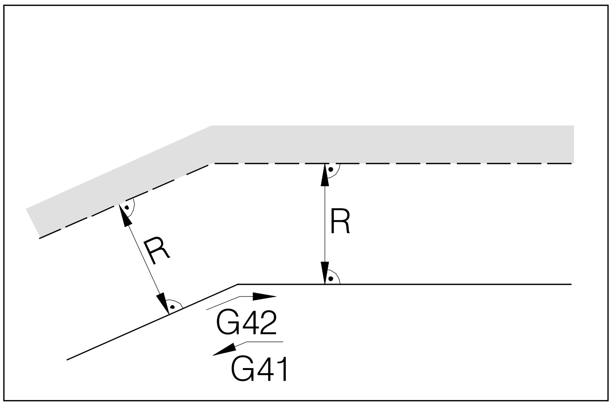
— — — programmed tool path
——— real traversed tool path

With arcs always the tangent of the end or start point of the arc will be approached.

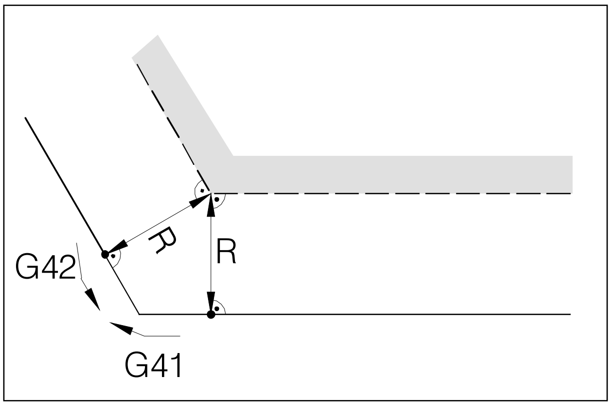
The approaching path to the contour and the leaving path from the contour must be larger than the tool radius R, otherwise program interruption with alarm.

If contour elements are smaller than the tool radius R, contour violations could happen. The software computes three blocks forward to recognize this contour violations and interrupt the program with an alarm.

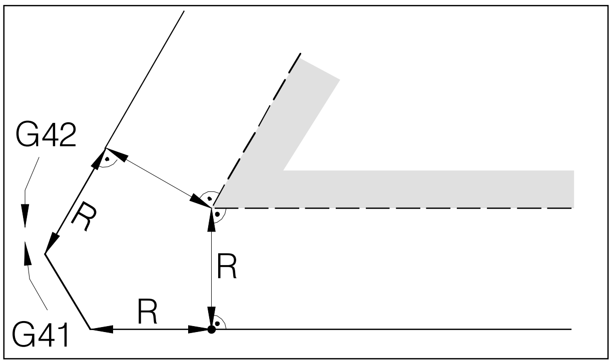
Tool pathes with program run with active cutter radius compensation



Tool path at an internal edge



Tool path at an outer edge >90°



Tool path at an outer edge <90°

— — — programmed tool path
——— real traversed tool path

With arcs always the tangent of the end or start point of the arc will be approached.

If contour elements are smaller than the tool radius R, contour violations could happen. The software computes three blocks forward to recognize this contour violations and interrupt the program with an alarm.

G48 Leave as Approached

Format
N... G48 X... Y... U...

To avoid cutting marks, a contour will be approached and left tangentially. To approach and leave the following functions are available:

G147 Soft Approach to Contour with Linear
G148 Soft Leaving with Linear
G247 Soft Approach to Cont. with Quarter Circle
G248 Soft Leaving with Quarter Circle
G347 Soft Approach to Contour with Semicircle
G348 Soft Leaving with Semicircle
see G147 - G348

With calling up G48, G41 or G42 must be active. G48 causes, that the contour will be left in the same way like it was approached.

G48 is effective blockwise.

After the G48 block G40 is active automatically.

G50 Cancel Scale Modification

G51 Scale Modification

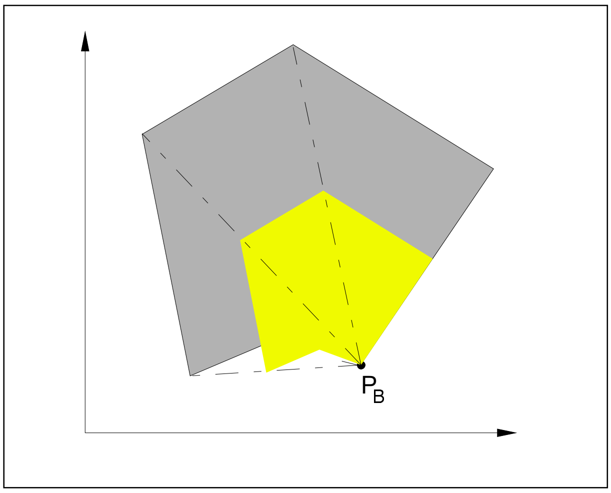
Format:
N... G50
N... G51 X... Y... Z... P...

With X, Y and Z the base point (P_B) for scale modiffication is determined, with P the scale factor.

If X, Y and Z are not determined, the work piece zero point (X = 0, Y = 0, Z = 0) is used as base point.

By scale modification the following values will be calculated:

- axis coordinates
- interpolation parameter
- radius / chamfer
- programmable zero offset



Enlarging a contour

G53 Cancel Zero Offset Blockwise

Format
N... G53

The machine zero is destinated by the machine manufacturer (EMCO milling machines: at the left front edge of the machine table).

Certain working sequences (tool change, measuring position...) are always proceeded at the same position in the working area.

With G53 all zero offsets but no tool offset will be suppressed for one block and all position data refer to the machine zero.

G54 - G57 Zero Offset 1 - 4 / Coordinate Rotation 1-4

Format
N... G54/G55/G56/G57

Four positions in the working area can be predetermined as zero points (e.g. points on fix mounted clamping devices).

The values of the zero offsets are stored in the setting data - ZERO OFFSET.

These zero offsets are called up with G54 - G59.

Additionally to the values of the zero offsets an angle for the coordinate rotation can be entered in the setting data - ROTAT. ANGLE.

These coordinate rotations are also called up with G54 - G59.

G54 is initial status and active without call-up.

G58/G59 Programmable Zero Offset / Coordinate Rotation

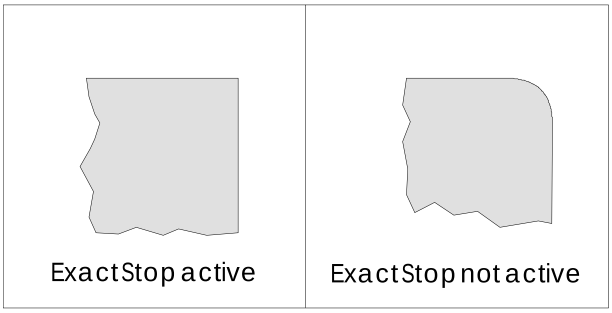
Format
N... G58/G59 A... X... Z...

These zero offsets will be added to the active zero offset G54 - G57.

With program end, program abortion and RESET G58/59 will be deleted.

With A a coordinate rotation angle can be programmed in degrees. This rotation also will be added to the active coordinate rotation of G54 - G57.

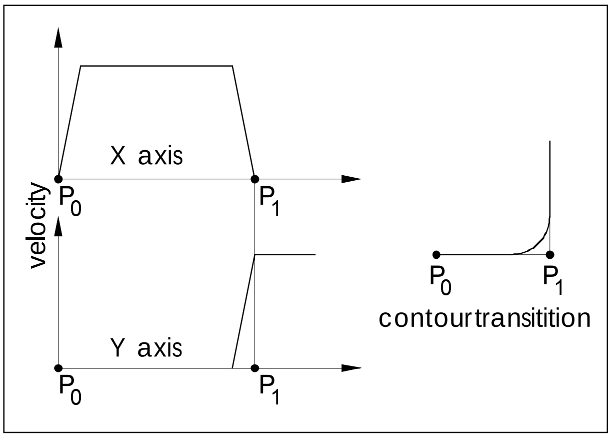
The commands G58 and G59 are effective blockwise, the zero offsets caused by these commands are active, until they will be cancelled or altered.



G60 Exact Stop Mode

Format
N... G60

A block will be worked off after the slides have been decelerated to standstill.
Edges will not be rounded and transitions will be exact.
G60 is effective until it will be deselected by G62 or G64.



Speed characteristics of the slides with G62/G64

G62,G64 Deselection Exact Stop Mode

Format
N... G62/64

Before the target point in X is reached, the Y axis will be accelerated. This enables steady movements.
The contour transitition is not exactly sharp-edged (parabola, hyperbola).
The size of the contour transitition is normally in the tolerance of the drawing.
With this software G62 and G64 have the same effect, but not with the SINUMERIK 810/820 M.

G70 Measuring in Inch

Format
N5 G70

By programming G70 the following inputs will be transformed into the inch system:
- Path information X, Y, Z
- Interpolation parameter I, J, K
- Chamfers, radii +U, -U

Note
- For clearness G70 should be defined in the first block of the program
- A change between G70 and G71 within a block is not allowed.
- A steady setting of the measuring system mm / inch will be proceeded in DIAGNOSIS, NC-MD. This setting is relevant for all values and will be kept also with power off / on.

G71 Measuring in Millimeter

Format
N5 G71

Comment and note like G70!

G80 Delete G81 bis G89
G81 Call Cycle L81
G82 Call Cycle L82
G83 Call Cycle L83
G84 Call Cycle L84
G85 Call Cycle L85
G86 Call Cycle L86
G87 Call Cycle L87
G88 Call Cycle L88
G89 Call Cycle L89

With G81 - G89 the cycles L81 - L89 will be called up. With the call up by a G command the cycles are modal, that means the cycles will be proceeded after every traverse movement, until they will be cancelled with G80.

Explanations see L81 - L89.

G90 Absolute Programming

Format
N... G90

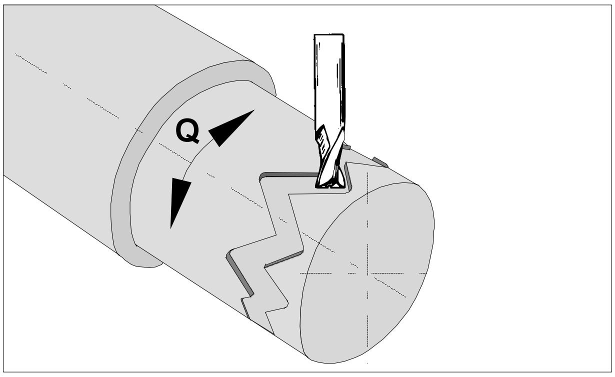
Notes
- Direct switchover within a block between G90 and G91 is not permitted.
- G90 and G91 may also be programmed with some other G functions
  (N... G90 G00 X... Z...).

G91 Incremental Programming

Format
N... G91

Notes see G90.

G 92 Cylindrical interpolation



Format
N... G92 P.. Q

G92 P.. ..... Cylindrical interpolation ON
G92 P1 ..... Cylindrical interpolation OFF
P .... Factor for unit circle
Q .... Axis name for rotary axis

Cylindrical interpolation permits machining of cylindrical paths with one rotary axis and one linear axis. Both linear and circular contours may be programmed. It is not possible to input the interpolation parameters I, J and K.
The position of the rotary axis is entered in degrees. The ration is programmed under G92 P... for this purpose.

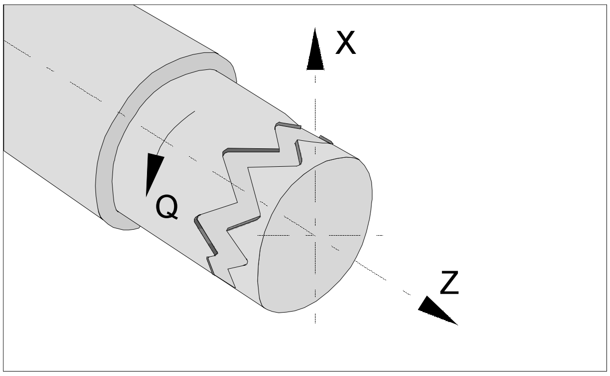
The control forms the ration from the machining diameter and the unit diameter as follows:

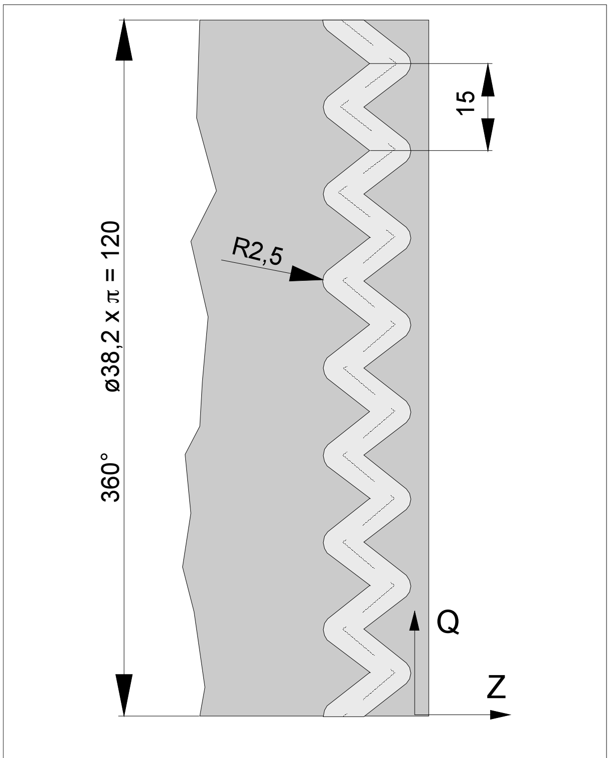
$$P= \frac{\text{machining diameter}}{\text{unit diameter}}$$

The unit diameter is derived from the ralation
p x π = 360 ab

$$\text{Unit diameter} = \frac{360}{\pi} \quad \text{in mm or inch}$$

No characters other than axis name must be written in a block containing G92P...





Milled with end milling cutter ø5mm

Example Cylindrical interpolation

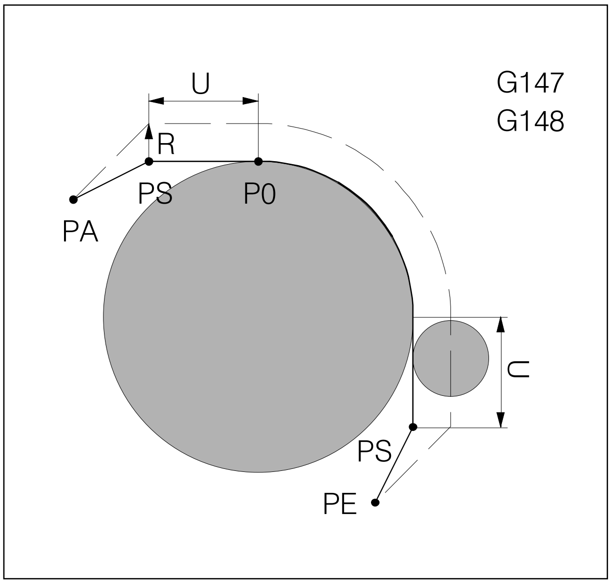
G54
G58 Z40
T7 D7
F200 S2=2000 G94 M2=3
M52 Couple in and Positioning of the spindle
G92 P0.33 Q Cylindrical interpolation ON.
Unit diameter = 360/PI = 114.59
P =38.2 /114.59= 0.33
Q = Axis name for round axis
G0 X45 Z-5
G1 X35 Q0 Z-5
G1 Z-15 Q22.5
Z-5 Q45
Z-15 Q67.5
Z-5 Q90
Z-15 Q112.5
Z-5 Q135
Z-15 Q157.5
Z-5 Q180
Z-15 Q202.5
Z-5 Q225
Z-15 Q247.5
Z-5 Q270
Z-15 Q292.5
Z-5 Q315
Z-15 Q337.5
Z-5 Q360
X45
G92 P1 Cylindrical interpolation OFF
M53 Q- axis OFF
G0 X80 Z100
M2=5 Spindle for driven tool OFF
M30

G94 Feed Rate in Minutes

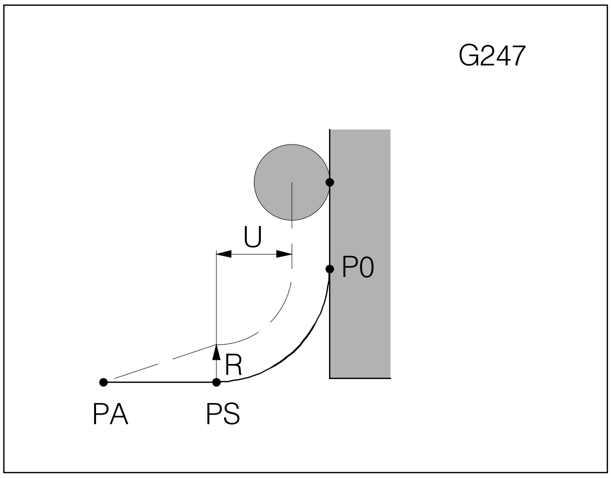
The entry of the command G94 means that all values programmed under "F" (feed) are in mm/min (inch/min).

G95 Feed Rate in Revolutions

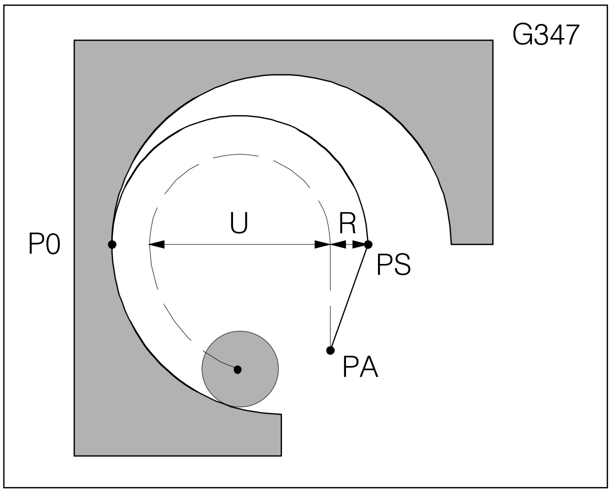
The entry of the command G95 means that all commands programmed under "F" are in mm/revolution (inch/revolution).



Soft approach and leaving with linear



Soft approach with quarter circle



Soft approach with semicircle

G147 Soft Approach to Contour with Linear
G247 Soft Approach to Contour with Quarter Circle
G347 Soft Approach to Contour with Semicircle
G148 Soft Leaving the Contour with Linear
G248 Soft Leaving the Contour with Quarter Circle
G348 Soft Leaving the Contour with Semicircle

Format
N... G147/247/347/148/248/348 X... Y... U...

- The functions for approaching and leaving a contour are effective blockwise.
- In the approaching block have to be determined: the coordinates of the start point P0 of the contour and the value U (approaching distance without contourtouch).
- In the leaving block have to be determined: the coordinates of the end point PE after leaving the contour and the value U (leaving distance without contourtouch)
- In an approaching or leaving block no further traverse movements must be programmed.
- After an approaching or leaving block no block with only miscellaneous functions is allowed.
- Before an approaching block G41 or G42 must be active.
- In the leaving block G40 will be set automatically, that means, G41 or G42 must be programmed again.
- Soft approaching or leaving is not possible with contours programmed with contour definition.

PA Start point before approaching to the contour

PS Supporting point, will be calculated by the control basing on U

P0 End point of the approaching block=start point of the contour

PE End point after leaving the contour

R Tool radius

U Distance without contourtouch

dashed line: path of the milling tool centre

Description of M Commands

M00 Programmed Stop

This command effects a machining stop within a part program.
The milling spindle, feeds and coolant will be switched off.
The machine door can be opened without releasing an alarm.

With "NC START" the program run can be continued. After that the main drive will be switched on with all values which were valid before.

M01 Programmed Stop, Conditional

M01 works like M00, when OPT. STOP is active (display OPT in the first line at the screen). If OPT. STOP is not active, M01 has no effect.

With "NC START" the program run can be continued. After that the main drive will be switched on with all values which were valid before.

M02 Main Program End

M02 works like M30.

M03 Milling Spindle ON Clockwise

The spindle will be switched on provided that a cutting speed has been programmed, the machine doors are closed and a workpiece is correctly clamped. M03 must be used for all right hand cutting tools.

M04 Milling Spindle ON Counterclockwise

The same conditions as described under M03 apply here.
M04 must be used for all left hand cutting tools.

M05 Milling Spindle OFF

The main drive is braked electrically.
At the program end the milling spindle is automatically switched off.

M06 Tool Change

only for machines with tool turret
The by T previously selected tool will be swivelled in.

After that the main drive will be switched on with all values which were valid before.

M08 Coolant ON

Only for EMCO PC Mill 100/125/155.
The coolant will be switched on.

M09 Coolant OFF

Only for EMCO PC Mill 100/125/155.
The coolant will be switched off.

M17 Subroutine End

M17 will be written in the last block of a subroutine. It can stand alone in this block or with other functions. The call-up of a subroutine and M17 must not stand in the same block (nesting).

M27 Swivel Dividing Head

Only for accessory dividing head.
The dividing head will be swivelled for one step (step angle mechanically adjusted).

M30 Main Program End

With M02 all drives will be switched off and the control will be resetted to program start.

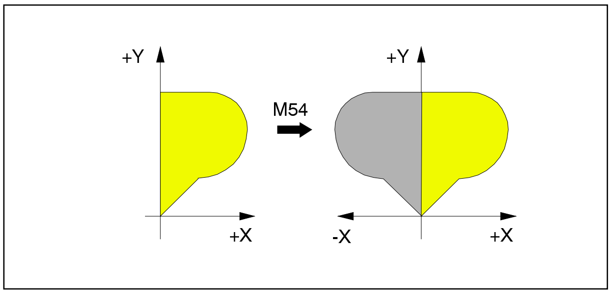
M53 - M58 Mirror Functions

Mirroring occurs around the work piece zero point.

Example program structure:

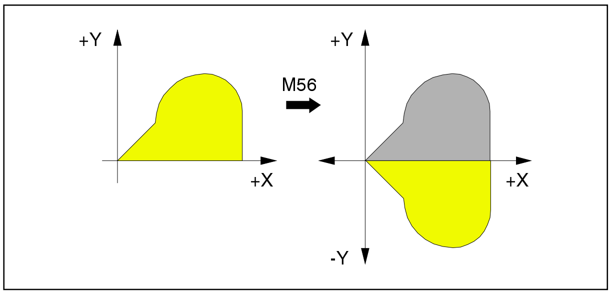
N... M54 L999
N... L123 P1
N... M53 L999
... The contour in subroutine L123 will be mirrored in X (around the Y axis).

In the block with M53 - M58 the cycle call L999 is necessary immediately, so that the PLC can change to the mirror mode, before the NC control goes on with the program run.



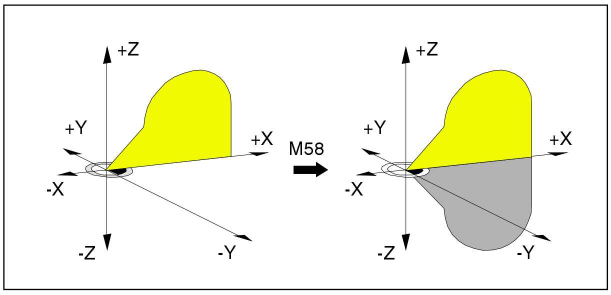
Mirroring the X values

Caution
If L999 is not programmed, the NC control is before the PLC in chronological order (no alarm). That means, that mirroring would start some blocks (depending on buffer memory) after M54/M56/M58 and would end some blocks after M53/M55/M57.
Uncontrolled movements!
Danger of collisions!



Mirroring the Y values

Mirroring in several axes is possible.
If mirroring occurs in one axis in the active plane, in the mirrored picture the circle directions (G02, G03) and correction directions (G41, G42) will be reversed.
With mirroring in both axes no reversation occurs.
Mirroring will not be displayed in graphic simulation.



Mirroring the Z values

M53 No mirror axis X
M54 Mirror axis X

M55 No mirror axis Y
M56 Mirror axis Y

M57 No mirror axis Z
M58 Mirror axis Z

M71 Puff Blowing ON

only for accessory blow off device.
The blow off device will be switched on.

M72 Puff Blowing OFF

only for accessory blow off device.
The blow off device will be switched off.

Description of Cycles

Cycles will be programmed in program in a manner, that first the R parameter will be written into the program ant then the cycle call with the number of cycle runs.

Example

N... R00=... R01=... R02=... R03=... R04=...
R05=... R10=... R11=... L83 P2

That means that the cycle L83 with the programmed parameter will run 2 times.

Note:

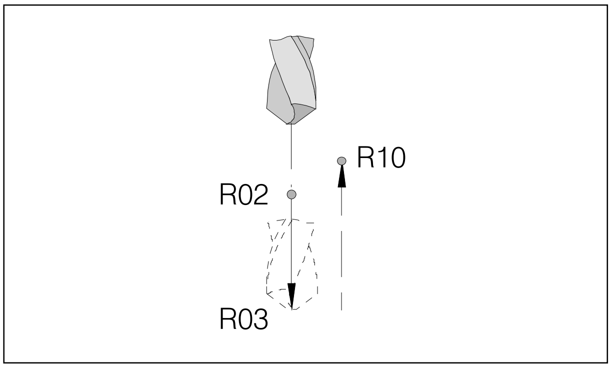
- Before cycle call a tool offset has to be active.
- The fitting feed,spindle speed and spindle turning direction have to be programmed in the part program before cycle call (expecting the cycles,in which these values can be programmed as R parameter).

With drilling cycles, which are called by G81 - G89, the variable values can be set with the parameter R00 - R17.
The hole position in the active plane has to be approached by the cycle-calling program.
The cycle called by G81 - G89 will be proceeded after every traverse movement, until it is deselected by G80.

Programming the parameter for G81 - G89:

R00	Dwell time at start point (chip removal)
R01	First drilling depth without sign (incremental)
R02	Reference plane (absolute)
R03	Final depth of hole
R04	Dwell at hole bottom (chip breaking)
R05	Amount of degression (incremental)
R06	Direction of rotation for retraction (M03, M04)
R07	Direction of spindle rotation (M03, M04)
R08	Thread tapping with/without encoder
R09	Thread pitch (only with tapping with encoder)
R10	Retraction plane(absolute)
R11	Deep-hole drilling with chip breaking or removal (L83)
R11	Number of drilling axis
R12	Retraction path horizontal with sign (incremental)
R13	Retraction path vertical with sign (incremental)
R16	Feed rate
R17	Retraction feed rate

L81 Drilling, Centering



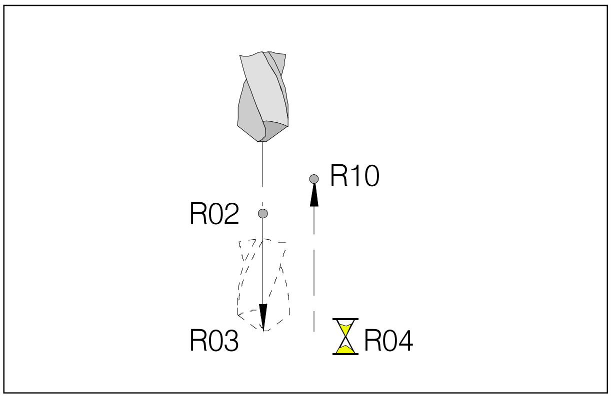
Movements of drilling centering cycle L81

The drilling tool traverses with rapid traverse to the reference plane (R02), with the previous programmed feed rate to the final depth (R03) and immediately back to retraction plane (R10) with rapid speed.

Programming the parameter

R02 Reference plane (absolute)
R03 Final depth of hole
R10 Retraction plane (absolute)

L82 Drilling, Spot Facing



Movements of drilling, spot facing cycle L82

The drilling tool traverses with rapid traverse to the reference plane (R02), with the previous programmed feed rate to the final depth (R03), dwells (R04) and retracts to retraction plane (R10) with rapid speed.

Programming the parameter

R02 Reference plane (absolute)
R03 Final depth of hole
R04 Dwell
R10 Retraction plane (absolute)

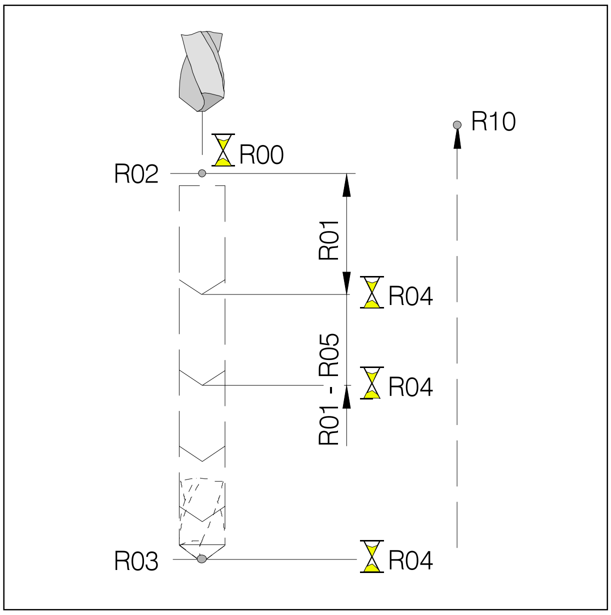
L83 Deep-hole Drilling

This cycle is for drilling deep holes or for drilling in materials with bad cutting properties.
With the parameter R11 the retraction movements can be determined:

Chip break (R11 = 0)
The drill dives into the work piece down to the first drilling depth (R01), dwells (R04), retracts 1 mm and dives in again.

Chip removal (R11 = 1)
The drill dives into the work piece down to the first drilling depth (R01), dwells (R04), retracts complete out of the boring (onto R02), dwells (R00) and dives in again.

The following infeed is each time for R05 shorter than the previous. The sequence infeed-retraction will be repeated, until final depth is reached.
If the infeed remains under R05 (by way of calculation), it will kept constant at the value of R05.
If the remaining infeed rest to the final depth R03 is less than the double amount of degression (2 x R05), the rest of infeed will be halved and worked off in two infeeds. So the smallest infeed never can be below R05/2.

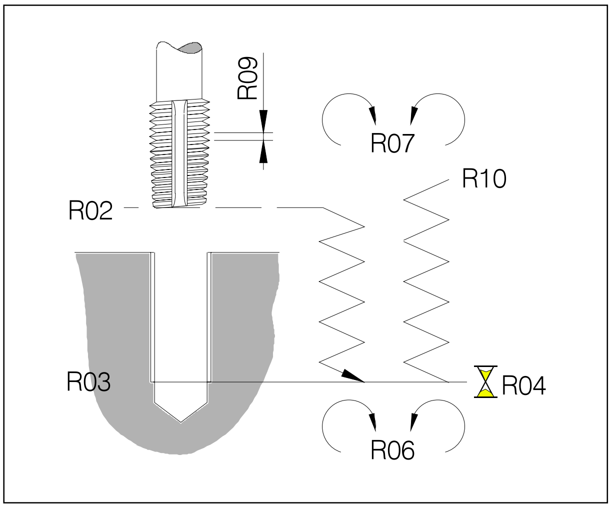


Movements of deep-hole drilling cycle L83

Programming parameter

R00	Dwell time at start point (chip removal)	
R01	First drilling depth without sign (incremental)	
R02	Reference plane (absolute)	
R03	Final depth of hole	
R04	Dwell at hole bottom (chip breaking)	
R05	Amount of degression (incremental)	
R10	Retraction plane (absolute)	
R11	0	chip break
	1	chip removal

L84 Thread Tapping with/without Encoder



Movements of Thread tapping cycle L84

The cycle L84 allows thread tapping with and without encoder on the milling spindle.
In both cases a **tapping chuck with length compensation** must be used.
Spindle override and **feed override must be set** to **100%**.

Programming the parameter

R02 Reference plane (absolute)
R03 Final depth of hole
R04 Dwell at thread depth
R06 Direction of rotation for retraction (M03, M04)
R07 Direction of rotation after cycle (M03, M04)
R08 Thread tapping -3=with, -4=without encoder
R09 Thread pitch (only with tapping with encoder)
R10 Retraction plane (absolute)
R11 Number of drilling axis

Notes to the parameter

R04 Dwell at thread depth:
The dwell is only effective without encoder.

R06 Direction of rotation for retraction:
R06=0: automatic reversing of spindle direction
For machines without encoder R06 has to be programmed.

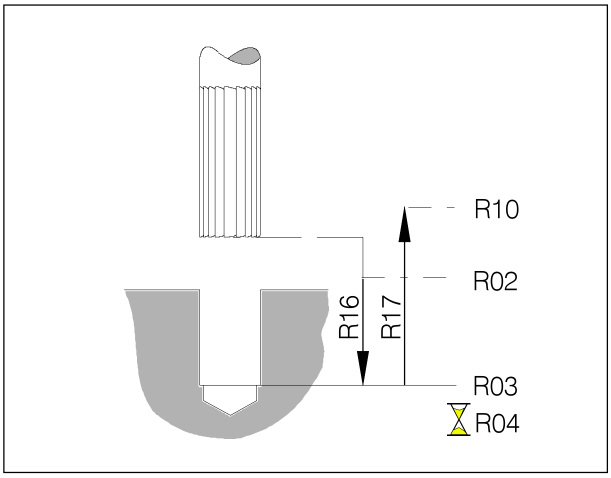
R07 Direction of rotation after cycle:
If the thread tapping cycle is called up with G84, the cycle needs for execution of the next tappings a rotation direction. This is programmed with R07.
If R06 = 0 was programmed, R07 may not be programmed any more.

R08 Thread tapping with/without encoder:
If a thread should be tapped without encoder, though an encoder is mounted at the machine, R08=-4 has to be programmed.
On machines without encoder R08 will be ignored.

R09 Thread pitch:
Thread pitch is only effective with encoder for calculation of the feed rate basing on spindle speed.
On machines without encoder a feed rate has to be programmed previousely in the part program.

R11 Number of drilling axis:
With R11 the number of the drilling axis may be programmed; if R11 was not programmed the drilling axis will be recognized through the active selected plane.

L85 Boring 1



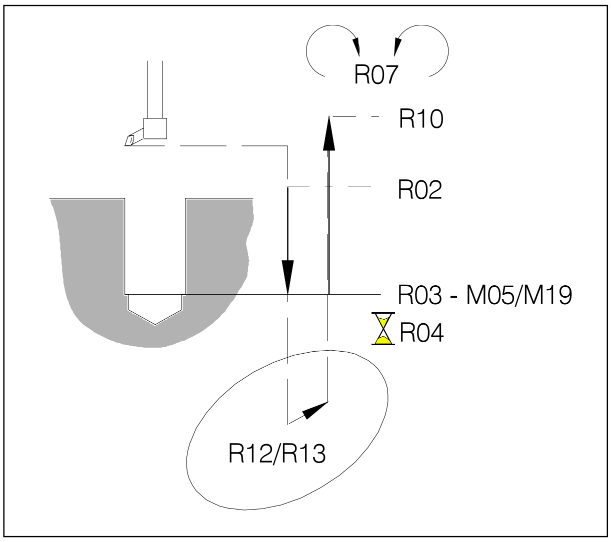
Boring cycle 1 L85

The tool traverses with rapid traverse to the reference plane (R02), with feed (R16) to the final depth (R03), dwells (R04) and retracts with retraction feed (R17) to the retraction plane (R10).

Programming the parameter

R02 Reference plane (absolute)
R03 Final depth of hole
R04 Dwell
R10 Retraction plane (absolute)
R16 Feed rate
R17 Retraction feed rate

L86 Boring 2



Boring cycle 2 L86

(only for machines with oriented spindle stop)

This cycle is for boring with boring and facing heads. After reaching the final depth the milling spindle will be stopped, the boring and facing head will be retracted from the surface (horizontal and vertical) and retraction occurs without touching the machined surface.

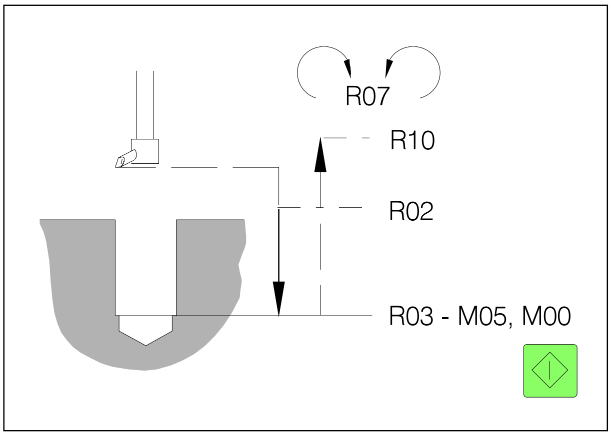
The tool traverses with rapid traverse to the reference plane (R02), with the feed rate determined in the part program to the final depth (R03), dwells (R04), the milling spindle stops, the tool traverses in rapid traverse horizontal (R12) and vertical (R13) away from the surface and traverses with rapid feed to the retraction plane (R10).

On machines with encoder occurs at the final depth an oriented spindle stop (M19). The angle can be set in the SETTING DATA - SPINDLE.
On machines without encoder occurs a spindle stop without orientation (M05).

Programming the parameter

R02 Reference plane (absolute)
R03 Final depth of hole
R04 Dwell
R07 Direction of spindle rotation (M03, M04)
R10 Retraction plane (absolute)
R12 Retraction path (horizontal with sign) (incremental)
R13 Retraction path (vertical with sign) (incremental)

L87 Boring 3



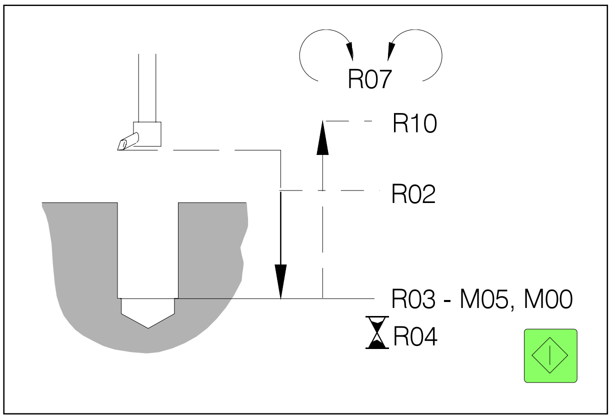
Boring cycle 3 L87

The tool traverses with rapid traverse to the reference plane (R02), with the feed rate which was determined in the part program to final depth (R03). At final depth the spindle stops (M05) and the program stops (M00). With the key NC start the program run will be continued, the tool traverses with rapid traverse to the retraction plane (R10).

Programming the parameter

R02 Reference plane (absolute)
R03 Final depth of hole
R10 Retraction plane (absolute)
R16 Feed rate

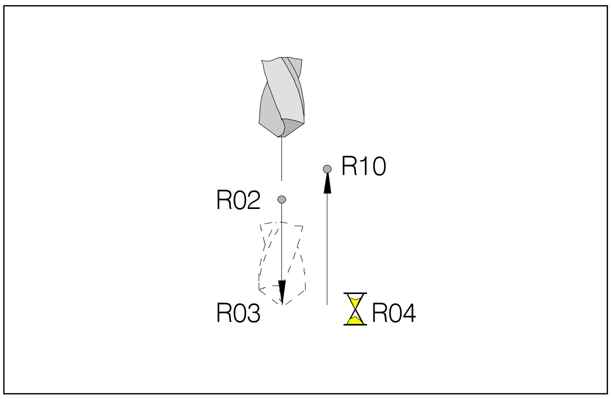
L88 Boring 4



Boring cycle 4 L88

This cycle is like L87, additionally a dwell at final depth can be programmed with R04.

L89 Boring 5



Boring cycle 5 L89

The tool traverses with rapid traverse to the reference plane (R02), with the previously programmed feed to the final depth (R03), dwells (R04) and retracts with feed back to the retraction plane (R10).

Programming the parameter

R02 Reference plane (absolute)
R03 Final depth of hole
R04 Dwell at final depth
R10 Retraction plane (absolute)

L96 Cycle for Tool Change

This cycle will be called without parameter.
This cycle includes all necessary commands (depending on the machine) for the tool change.
This cycle is included in the software package for every machine.

Drilling and Milling Patterns

Following parameter are used in the cycles L900 - L930:

Programming the parameter for L900 - L930

R01 Infeed depth (incremental)

R02 Reference plane (absolute)

R03 Depth (slot-, pocket-, elongated hole-, circular slot-) (absolute)

R06 Milling direction (G03, G04)

R10 Retraction plane (absolute)

R12 Pocket length (incremental)

R12 Slot width (incremental)

R13 Pocket width (incremental)

R13 Length (slot, elongated hole, angle for slot length) (incremental)

R15 Feed rate (pocket surface)

R16 Feed rate (pocket depth)

R22 Centre point... (horizontal)

R23 Centre point... (vertical)

R24 Radius (corner-, pocket-)

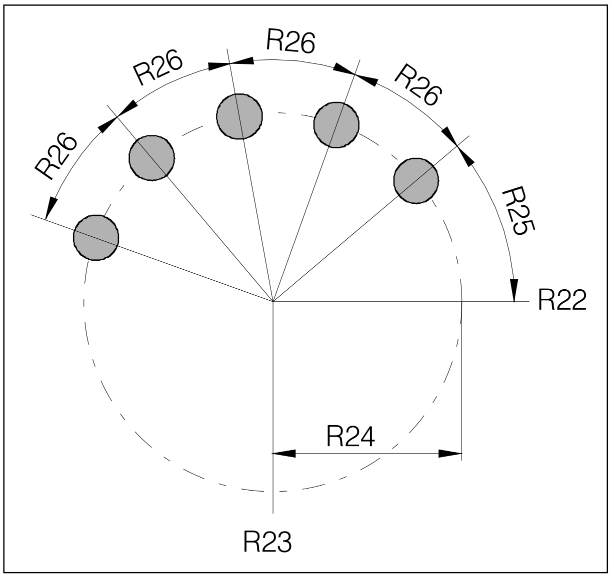
R25 Starting angle

R26 Indexing angle

R27 Number of slots, holes, elongated holes

R28 Number of drilling cycle (L81 - L89)

L900 Drilling Pattern Hole Circle



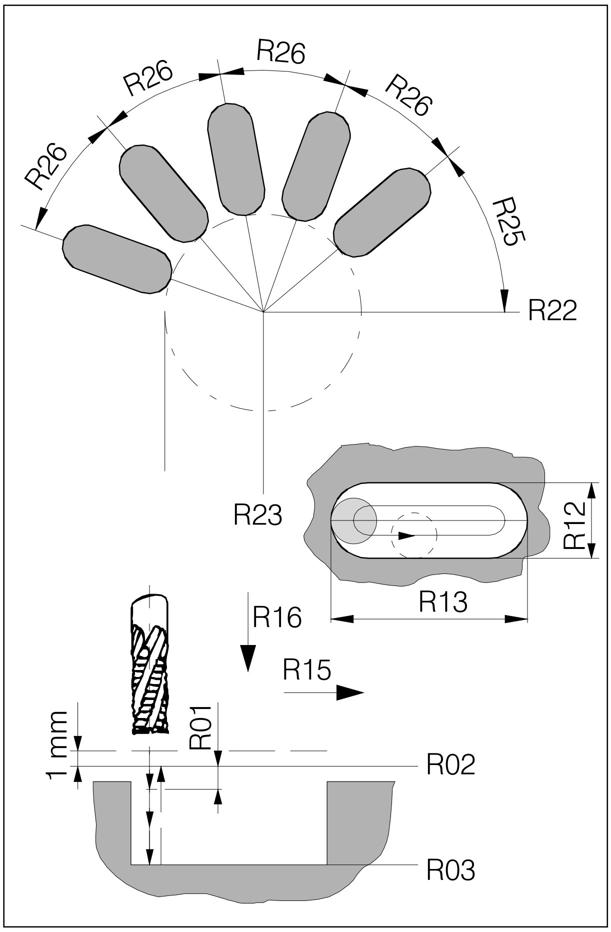
Drilling pattern L900

With L900 hole circles can be drilled. The single holes can be drilled with the cycles L81 - L89.
The parameter for the selected cycle L81 - L89 have to be programmed previously in the part program.
The cycle works in the active plane.

Programming the parameter

R22 Centre point of the hole circle (horizontal, absolute)
R23 Centre point of the hole circle (vertical, absolute)
R24 Radius of the hole circle
R25 Starting angle, related to the horizontal axis
R26 Indexing angle
R27 Number of holes
R28 Number of the drilling cycle to be executed (L81 - L89)

L901 Milling Pattern Slot

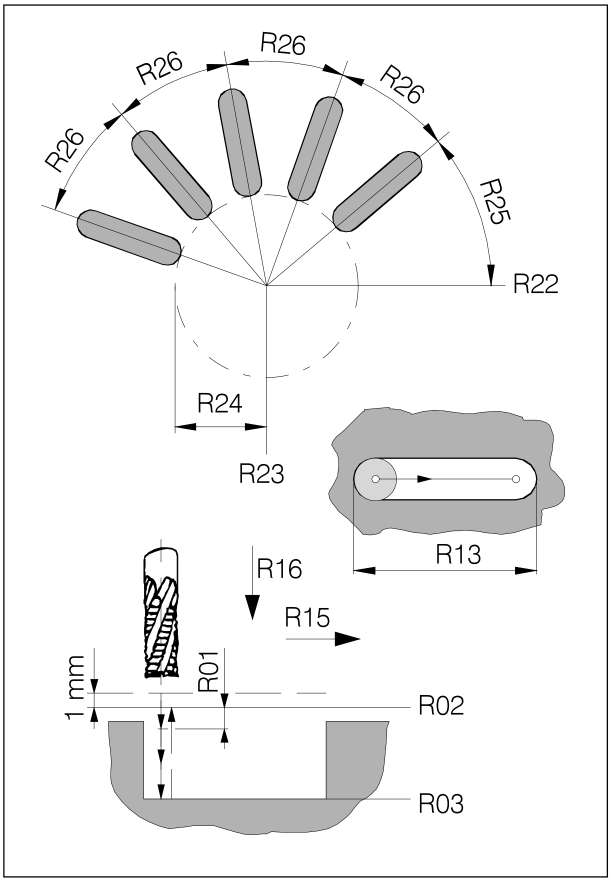


Milling pattern slot L901

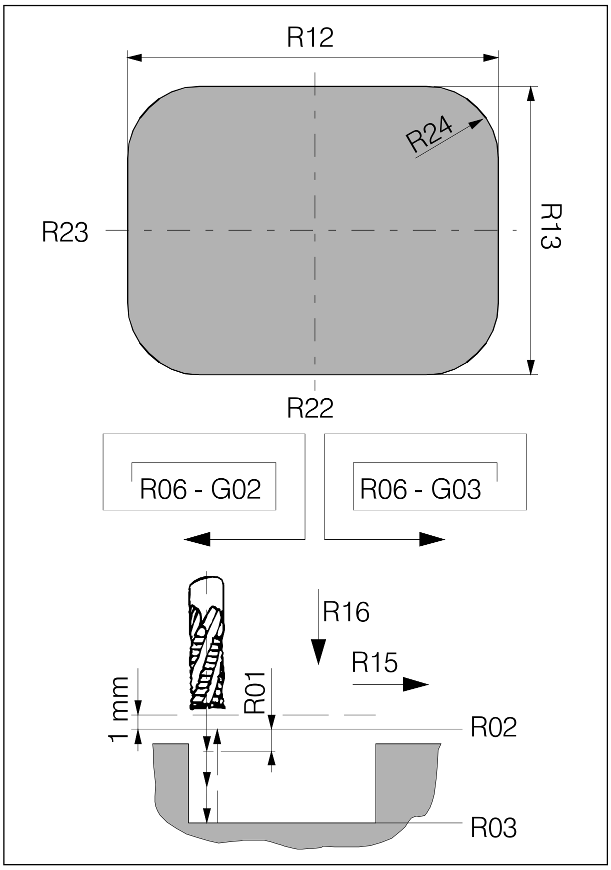
With L901 slot circles can be milled.
The cycle selects and deselects the cutter radius compensation automatically. The tool movement in the slot is counterclockwise.
The cycle works in the active plane.

Programming the parameter

R01 Infeed depth (incremental, without sign)
With R01=0 the whole depth will be fed in at once. With a rest depth less than 2 x R01 the rest will be worked off in 2 equal infeeds.
R02 Reference plane
R03 Slot depth
R12 Slot width
The tool diameter must be less than 0.9 x slot width and more than 0.5 x slot width
R13 Slot length
R15 Feed rate in length direction
R16 Feed rate in infeed direction
R22 Centre point circle of slots (horizontal, absolute)
R23 Centre point circle of slots (vertical, absolute)
R24 Radius of the circle of slots
R25 Starting angle, related to the horizontal axis
R26 Indexing angle
R27 Number of slots



Milling pattern elongated hole L902



Rectangular pocket L903

L902 Milling Pattern Elongated Hole

With L902 circles of elongated holes can be milled.
This cycle works without cutter radius compensation.
The tool diameter determines the with of the elongated hole.
The cycle works in the active plane.

Programming the arameter

R01 Infeed depth (incremental, without sign) see L901
R02 Reference plane
R03 Elongated hole depth
R13 Elongated hole length
R15 Feed in longitudinal direction
R16 Feed in infeed direction
R22 Centre point elongated hole circle (horizontal, absolute)
R23 Centre point elongated hole circle (vertical, absolute)
R24 Radius of the elongated hole circle
R25 Starting angle, related to the horizontal axis
R26 Indexing angle
R27 Number of elongated holes

L903 Milling Rectangular Pocket

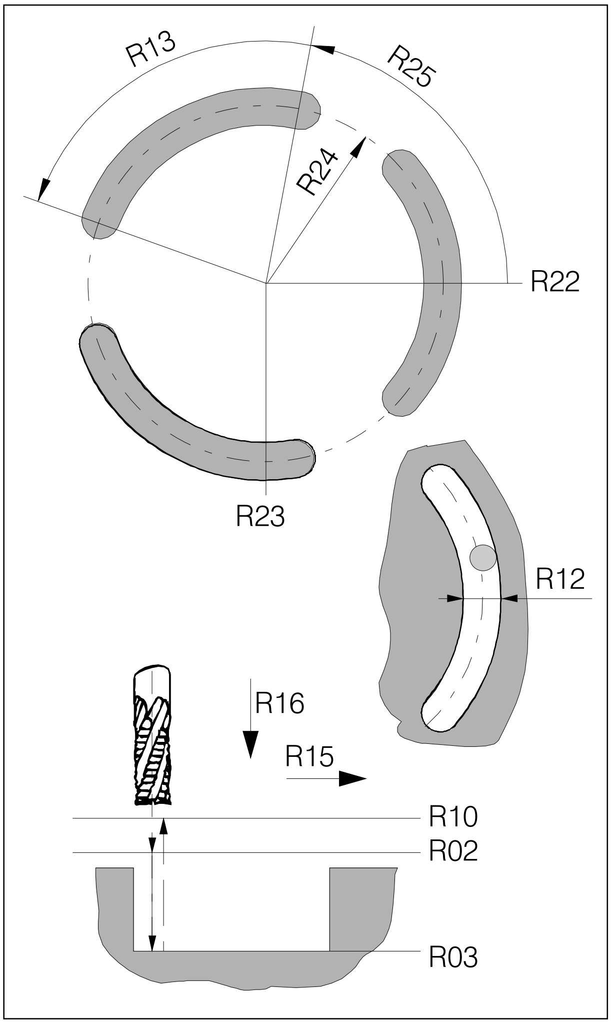
With L903 rectangular pockets can be milled.
Cutter radius compensation will be deselected automatically, the tool radius will be considered with the values from the tool data register
The cycle works in the active plane

Programming the parameter

R01 Infeed depth (incremental, without sign) see L901.
R02 Reference plane
R03 Pocket depth
R12 Pocket length
R13 Pocket width
The radius of the milling cutter has to be less than the half of the smaller pocked side.
R15 Feed rate in length direction
R16 Feed rate in infeed direction
R22 Centre point of the pocket (horizontal, absolute)
R23 Centre point of the pocket (vertical, absolute)
R24 Corner radius of the pocket
The corner radius has to be equal or larger than the cutter radius.

L904 Milling Pattern Circular Slot

With L904 circular slots can be milled.
The cycle selects and deselects the cutter radius compensation automatically.
The cycle works in the active plane.



Milling pattern circular slot L904

Programming the parameter

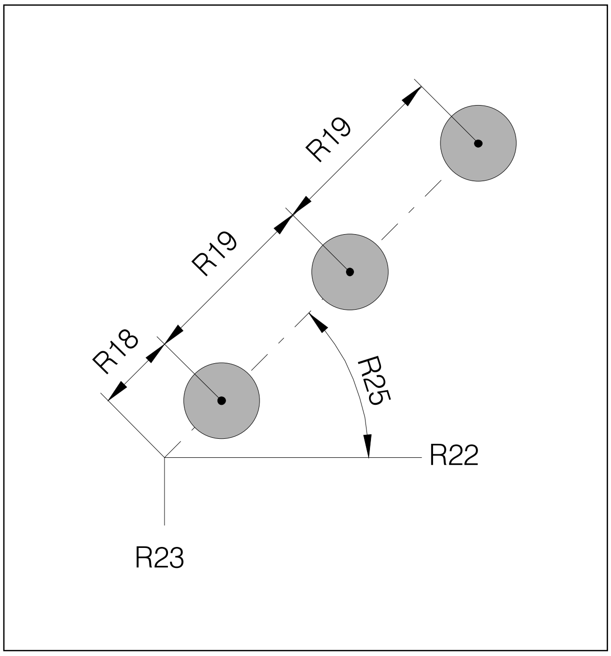
R02 Reference plane
R03 Circle slot depth
R12 Circle slot width
The tool diameter must be less than 0.9 x slot width and more than 0.5 x slot width
R10 Retraction height
R13 Angle for slot length, related to the horizontal axis
R15 Feed in slot direction
R16 Feed in infeed direction
R22 Centre point circular slot (horizontal, absolute)
R23 Centre point circular slot (vertical, absolute)
R24 Radius of the slot circle
R25 Starting angle, related to the horizontal axis
R27 Number of circular slots

L905 Drilling Pattern Single Hole

With L905 a single hole can be drilled with the desired cycle L81 - L89.
The parameter for the selected cycle L81 - L89 have to be programmed previously in the part program.
The cycle works in the active plane.

Programming the parameter

R22 Centre point hole (horizontal, absolute)
R23 Centre point hole (vertical, absolute)
R28 Number of the desired cycle
(L81 - L89)



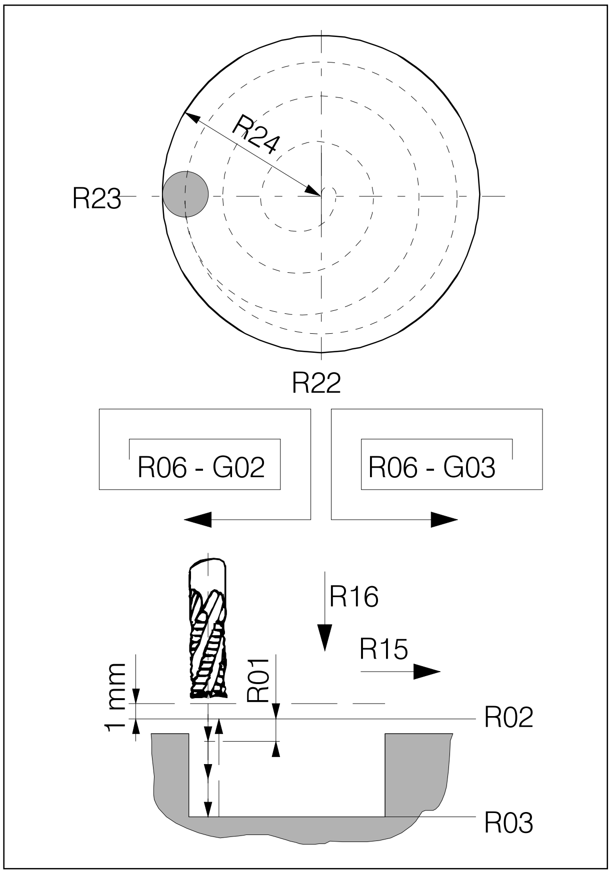
Row of holes L906

L906 Drilling Pattern Row of Holes

With L906 a row of holes with constant hole distance and any angle can be drilled.
For the drillings the cycles L81 - L89 can be used. The parameter for the selected cycle L81 - L89 have to be programmed previously in the part program.
The cycle works in the active plane.

Programming the parameter

R18 Distance from start point
R19 Hole distance
R22 Start point row of holes (horizontal, absolute)
R23 Start point row of holes (vertical, absolute)
R25 Angle of the row of holes, related to the horizontal axis
R27 Number of holes
R28 Number of the drilling cycle (L81 - L89)



Circular pocket L930

L930 Milling Pattern Circular Pocket

With L930 a circular pocket can be milled.
The cycle deselects the cutter radius compensation automatically.
The tool radius will be considered with the radius entered in the tool offset register.
The cycle works in the active plane.

Programming the parameter

R01 Infeed depth (incremental, without sign) see L901
R02 reference plane
R03 Pocket depth
R06 Milling direction (G02/G03)
The tool traverses after infeed a spiral path from the centre outwards. The milling direction (climb or upcut milling) has to be programmed with R06 = 02/03.
R15 Feed rate in milling direction
R16 Feed rate in infeed direction
R22 Centre point pocket (horizontal, absolute)
R23 Centre point pocket (vertical, absolute)
R24 Radius of the circular pocket

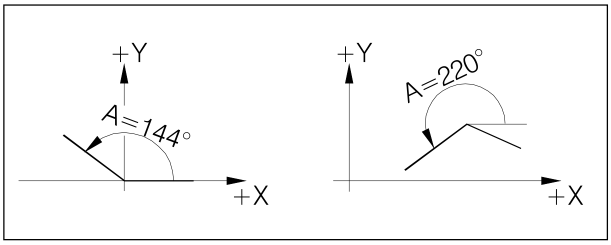
L999 Clear Buffer Memory

The cycle L999 will be called without parameters.
The program interpreter (part of the software, which reads in and works off block by block) stops reading in and works off the block in the buffer memory, until the buffer memory is empty.
After that the program interpreter starts reading in and workong off new blocks.

This cycle is used with mirroring to gain time for the PLC to switch over to mirror mode, before the blocks to be mirrored will be worked off.

Contour Definition

This way of programming can be used if intersection point coordinates are missing. Multiple point drafts for contour definition are offered in different forms and may be combined at will. Intersection points will be calculated by the software from coordinate values or angles.
The following pictures are related to G17 (X-Y plane active).



Angles refer to the +X direction

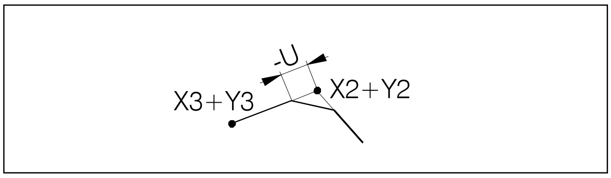
Angles are always relatet to the +X direction

The following contour drafts are offered:

Insert Chamfer

With chamfers U is entered as a negative number.

G1 X2... Y2... U-... LF
G1 X3... Y3... LF

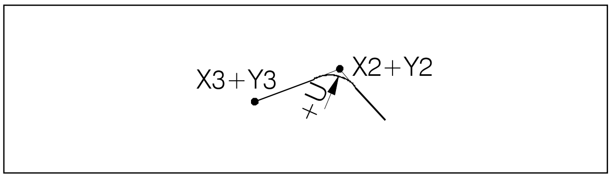


Inserting a chamfer

Insert Radius

With radii U is entered as a positive number. The inserted radius must be smaller than the shorter one of the two lines.

G1 X2... Y2... U+... LF
G1 X3... Y3... LF

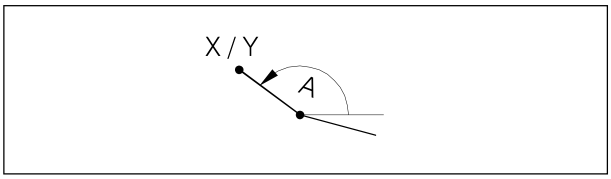


Inserting a radius

Line

By entering the angle A and one target point coordinate the line will be computed.

G1 A... X... or G1 A... Y...

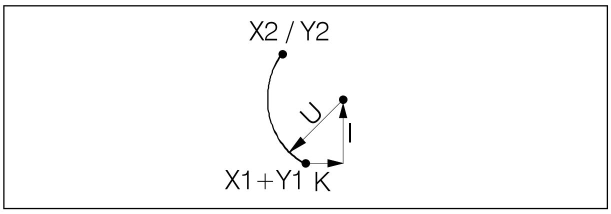


Line determined by angle and one end coordinate

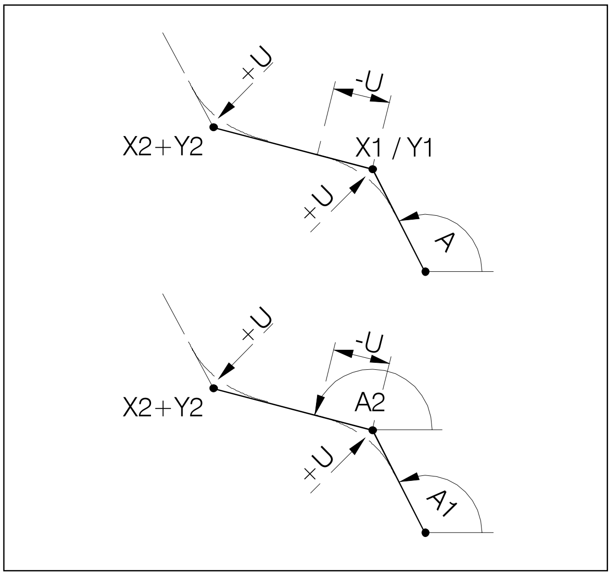
Arc

Description of the arc by radius U, centre point I, K and one target point coordinate.

G2 U... I... K... X2... or G2 U... I... K... Y2...



Arc defined by radius, centre and one end coordinate



Contour draft line - line

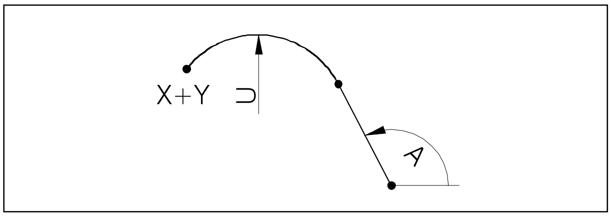
Line - Line

Angle A, one coordinate of the first point and both coordinate of the target points.
Radii +U or chamfers -U can be inserted, if the next following block is a G1 block, they also can be added on.

G1 A... X1... X2... Y2... or G1 A... Y1... X2... Y2...

Second possibility:
Angles A1 and A2, both coordinates of the target point.

G1 A1... A2... X2... Y2...

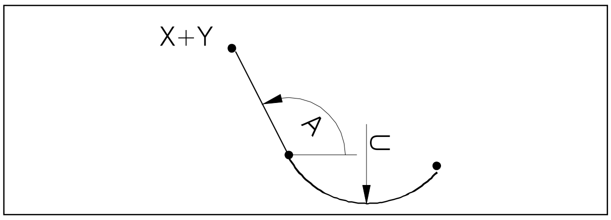


Contour draft line - arc (tangential)

Line - Arc (tangential)

Angle A, radius U and both coordinates of the target point.

G3 A... U... X... Y...

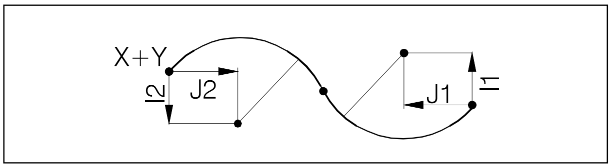


Contour draft arc - line (tangential)

Arc - Line (tangential)

Radius U, angle A and both coordinates of the target point.

G2 U... A... X... Y...



Contour draft arc - arc (tangential)

Arc - Arc (tangential)

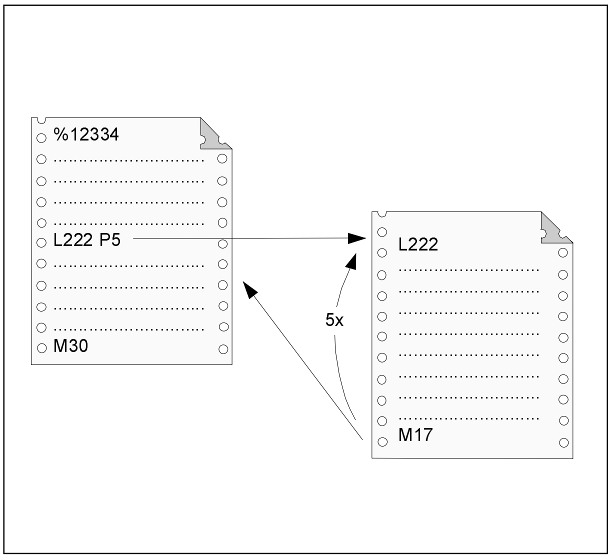
The path command G2, G3 will be programmed for the first arc. The second path command is opposite and will not be programmed. The interpolation parameters I2, K2 of the second arc are related to the end point of this arc. Both interpolation parameter have to be programmed, even if the value is 0.

G2 I1... K1... I2... K2... X... Y...

Subroutines

Functions which are repeated multiple can be programmed as subroutines.

The cycle numbers are reserved and must not be used for subroutines.



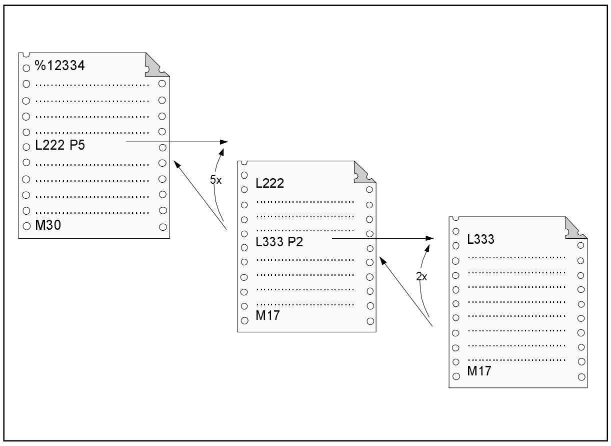
Program sequence with a subroutine

Subroutine Call in Part Program

e.g.: L123 P1 LF
L Subroutine
123 Subroutine number
P1 Number of subroutine runs (max. 99)

Subroutine End with M17

e.g.: N150 M17 LF



Subroutine nesting

Subroutine Nesting

A fourfold nesting of subroutines is possible. Block search is possible into the fourth subroutine level.

E: @-Codes

The CNC controls SINUMERIK 810 and SINUMERIK 820 can be programmed with @-comdes.
@-codes allow to program jumps in program, queries, branches etc.
The @-codes can be included in the main program or in subprograms that can be used e.g. as cycles.

Note:

For normal NC-programming the @-codes are almost not used, because the G and M commands and the standard cycles are enough.
You will need the @-codes when you want to design your own cycles with parameter interchange etc..

Programming @-codes is an advanced, complex way of programming. In the following the @-codes that work with WinNC are listed with a short description only.

Key

y → Relational operator <rop>
- 0: . . . . No condition
- 1: = ... equal to
- 2: <> . . . . Not equal to
- 3: > . . . . Greater than
- 4: >= . . . . Greater than or equal to
- 5: < . . . . Less than
- 6: <= . . . . Less than or equal to
- 7: . . . . True
- 8: . . . . Not

@- Code		Function
@040	<Const><R Par 1> ... <R Par n>	Saving of the entzered R parameters to stack
@041	<R Par 1><R Par 2>	Saving of a group of R parameters to stack
@042	<Const><R Par n> ... <R Par 1>	Fetching saved R parameters from stack
@043	<R Par 1><R Par 2>	Fetching a group of saved R parameters from stack
@100 @100	<Const> <R Par>**	Absolute jump to NC block
@111	<Var> <Value 1><Const 1> <Value 2><Const 2> ... ... <Value n><Const n>	Case branching
@12y	<Var><Value><Const>	IF-THEN-ELSE statement y Ñ relational operator <rop> Var Ñ R parameter or pointer
@13y	<Var><Value><Const>	Repeat statement with sampling of repeat condition at start y Ñ relational operator <rop>
@14y	<Var><Value><Const>	Repeat statement with sampling of repeat condition at end y Ñ relational operator <rop>
@151	<Var><Value 2><Const>	Repeat statement with repetitions until <Var> has incrementally reached <Value 2>
@161	<Var><Value2 2><Const>	Repeat statement with repetitions until <Var> has decrementally reached <Value 2>
@200	<Var>	Delete variable
@201	<Var><Value>	Load variable with vaue
@202	<Var 1><Var 2>	Swapping of variable contents
@210	<Value 3><Value 4>	Clear machine input buffer Value 3: MIB start address Value 4: MIB end address
@300	<Var><Value 1>	Machine data NC Value 1: Byte addresses 0 ... 4999
@302	<Var><Wert 1><Wert 2>	Machine data NC-Bits Value 1: Byte adresses 5000 ... 6999 Value 2: Bit adresses 0 ... 7
@310	<Var><Wert 1>	Setting data NC Value 1: Address 0 ... 4999
@320	<Var><Value 1><Value 2>	Tool offset Value 1: 0 Value 2: D no. 1 ... 99 Value 3: P no. 0 ... (9)
@330	<Value 3>	Settable zero offset (G54 - G57) Value 1: group 1 ... 4 (G54 - G57) Value 2: axis no.1, 2, ... Value 3: 0/1 (coarse/fine)
@331	<Var><Value 1><Value 2>	Progammable zero offset (G58, G59) Value 1: Goup 1 or 2 (G58 oder G59) Value 2: axis no.1, 2, ...
@336	<Value 3>	Total offset Value 2: axis no.1, 2, ...
@342	<Var><Value 1><Value 2>	Read programmed spindle speed Value 1: channel no. 0 ... 3 Value 2: spindle no. 1 ... 6

@- Code		Function
@345	<Var><Value 2>	Programmed cutting speed Value 1: channel no. 0, 1, 2 Value 2: 0 = G96
@360	<Var><Value 1><Value 3>	Actual axis position workpiece-related Value 2: axis no.1, 2, ...
@361	<Var><Value 1><Value 2>	Actual axis position machine-related Value 2: axis no.1, 2, ...
@363	<Var><Value 2>	Actual spindle position Value 2: spindle no. 1 ... 6
@364	<Var><Value 2>	Actual spindle speed Value 2: spindle no. 1 ... 6
@367	<Var><Value 1>	Read axis number of current plane/leading spindle into R parameter <Var>: Var+0: no. of horizontal axis Var+1: no. of vertical axis Var+2: no. of the axis perpendicular to the plane Var+3: no. of the axis in which length 2 is effective Var+4: no. of the leading spindle Value 1: channel no. 0, 1, 2
@36a	<Var><Value 1>	Actual D function Value 1 = 0
@36b	<Var><Value 1><Value 3>	Reading the G function of the current block out of the working memory Value 1: channel no. 0, 1, 2 Value 3: internal G group to which the G code belongs 0 ... 15
@371	<Var><Value 1><Value 3>	Special bits Value 1: channel no. 0 .. 2 = channel dependent, 99 = channel-independent
@3e4	<Var><Value 1>	Read active gear position Value 1: spindle no. 0 to 6
@420	<Wert1><Value 2><Value 3> <Value>	Tool offset Value 1: 0 Value 2: D no. 1 ... 99 Value 3: P no. 0 ... 7 (9)
@423	<Value 1><Value 2><Value 3> <Value>	Tool offset additive Value 1: 0 Value 2: D no. 1 ... 99 Value 3: P no. 0 ... 7 (9)
@430	<Value 1><Value 2><Value 3> <Value>	Settable zero offset Value 1: Group 1 ... 4 (G54 - G57) Value 2: axis no. 1, 2, ... Value 3: 0/1 (coarse/fine)
@431	<Value 1><Value 2><Value 3> <Value>	Settable zero offset additive Value 1: Group 1 ... 4 (G54 - G57) Value 2: axis no. 1, 2, ... Value 3: 0/1 (coarse/fine)
@432	<Value 1><Value 2><Value>	Programmable zero offset (G58, G59) Value 1: Group 1 or 2 (G58 oder G59) Value 2: axis no. 1, 2, ...
@440	<Value 3><Value>	Programmed axis position Value 3: axis no. 1, 2, ...
@442	<Value 3><Value>	Programmed spindle speed Value 3: spindle no. 0 to 6
@446	<Value>	Programmed radius
@447	<Value>	Programmed angle

@- Code		Function
@448	<Value 3><Value>	Programmed interpolation parameter for circle and thread Value 3: axis no. 1, 2, ...
@4e1	<Value 1><Value 2><Value>	Spindle acceleration time constant Value 1: spindle no. 0 to 6 Value 2: gear position1 to 8 Value 3: Spindle acceleration time constant 0 to 16000
	<Var> = <Value 1> + <Value 2> <Var> = <Value 1> - <Value 2> <Var> = <Value 1> x <Value 2> <Var> = <Value 1> / <Value 2>	Addition Subtraction Multiplication Division
@610	<Var><Value>	Absolute value generation
@613	<Var><Value>	Square root
@614	<Var><Value 1><Value 2>	Root from sum of squares
@620	<Var>	Incrementing of <Var> by 1
@621	<Var>	Decrementing of <Var> by 1
@622	<Var>	Integer
@630	<Var><Value>	Sine
@631	<Var><Value>	Cosine
@632	<Var><Value>	Tangent
@634	<Var><Value>	Arc Sine
@637	<Var><Value 1><Value 2>	Angle from two vector components
@640	<Var><Value>	Natural logarithm
@641	<Var><Value>	e Exponential function
@710	<Var 1><Var 2>	Reference processing Var 1: Output data from Var 1 Var 2: Input data from Var 2
@711	<Var 1><Var 2><Var 31>	Schnittpunktberechnung Var 1: Output data from Var 1 Var 2: First contour from Var 2 Var 3: Preset with 0
@713	<Var>	Start preparation for cycles Var: Output data from Var
@714		Stop of decoding util buffer is empty

G: Survey Pages

Survey Softkey Explanations

ACTUAL BLOCK
(Current block)
The current block is the block currently being processed. The display shows: The block **before** the current block, the current block and the block **after** the current block.

ADD. FUNCTION
(Additional function)
The additional functions (5th M Group) fixed by the machine tool manufacturer are grouped together in this function. Preselect: softkey "GUIDING"

AREA W-PIECE
(Work piece area)
You enter the simulation area, and the workpiece dimensions.

AXIAL
You enter the setting data for axes.

BLOCK END
You select the character for "Block End (LF)" via softkey.
Preselect: "GUIDING" softkey.

BLOCK NUMBER
The block number is automatically generated by the control in steps of five.

BLOCK SEARCH
Block search makes it possible to start the operation at any point in the program. During block search, the same calculations are carried out as in normal program operation, however there is no axis movement.

BTR START
not active

CIRCLE
You select the G functions for circular interpolation with this softkey.
Preselect: "GUIDING" softkey (operator prompting).

CONTOUR
This function simplifies programming of workpiece contours with the transfer of values directly from the drawing and graphic displays.
You can select the following elements and combinations:

- Line
- Line - circle
- Circle
- Circle - line
- Circle - circle
- 2 - point definition
- 2 - angle definition

COPY
You copy a part program and re-enter it into memory under another program number.

CORR. BLOCK
(Correction block)
An error in the program is marked with the cursor (correction pointer).

CURRENT PRG PTR
(current program pointer)
The display shows the subroutine nesting with number of runs and actual block number of the subroutine.

CURRENT VALUES
The display shows the values valid for the current operating sequence.

CYCLES
The following are displayed:

- The stored cycle numbers
- The number of characters used
- The free memory space

Cycles are protected subroutines which can be called for frequently used technologies or for machine-specific operations (stock removal cycles, drilling cycles, tool change cycles).
The values required are defined as parameter assignments before the cycles are called (cycle parameter assignment).

DATA IMPORT
Data import

DATA-IN START
(Start data input)
You start data input.

DATA IN-OUT
The data (part program, settable zero offsets, R parameters, tool offset) are read in from an external device (e.g. punched tape reader) or put out to an external device (e.g. printer).
Data input and output is handled via interface 1 or 2, drive A, B or C or printer.

DATA OUT
You select data output via the universal interface. You decide the data type with the subsequent softkeys. Using "START" (in "PART PROGRAM" with "MAINPRG. START" or "SUBROUT. START") you activate data output.

DEC-SBL YES-NO
(Decoding single block on-off)
With "YES", the blocks are processed singly. The function is activate at the end of the block in which decoding takes place with the signal present (program control).

DELETE
Using "DELETE" you clear one or more part programs in the program memory.

DELETE PRE-VER
not active

DIAGNOSIS
All current alarms are displayed separately as NC alarms, PLC alarms, PLC alarms and PLC messages. Other displays are for service purposes.

DIRECTORY
The following is displayed:

- The stored part program numbers
- The number of characters used
- The free memory space

DNC
Setting of the serial DNC interface.

DRF ASSIGNM
not active

DRF YES-NO
not avtive

DRY RUN YES-NO
With "YES", the axes are traversed at the dry run feedrate, not the programmed feedrate. The dry run feedrate is set via setting data.

EDIT
EDIT leads to "SELECT PROGRAM", "GUIDING" and "SIMULATION".
Editing means:
Input of a program into the memory or changing or altering a program already in the memory.

EXT START
Output of the end of block character (end of text)

EXT ZO
not active

FEED
You select the G functions for the type of feed via softkey. Preselect: softkey "GUIDING".

G FUNCT.
You select the G functions from groups G0 to G12 via softkeys.
Preselect: "GUIDING" softkey.

GUIDING
The operator guiding (operator prompting) speeds up and simplifies the input of part programs. Apart from geometric functions (G function, contour definition) you can also input machining cycles and technological functions (feedrate, spindle speed) via softkeys.

I/O
Setting the serial interface.

LINE
You select the G functions for threading via softkeys.

LIST OF TOOLS
The location number and tool number of worn tools is displayed.

MACH DATA
not active

MAINPRG. START
(Main program start)
You activate the output of part programs via the universal interface.

MAIN PROGRAM
The following is shown in the display which appears if you press this softkey:

- the numbers of the main programs stored
- the number of characters taken up in memory
- the free memory capacity.

After "DATA OUT" or "DATA IMPORT" you can select input or output of main programs with "MAIN PROGRAM".

MOVE
not active

NC ALARM
All current NC alarms are displayed.

NC MD
(NC machine data)
The NC machine data are displayed.

OPS
not active

OPT. STOP YES-NO
(Programmed stop ON-OFF)
With "Yes", the processing of the program is stopped at the point at which the "M01" command is programmed.

OVERR. YES-NO
not active

OVERSTORE
You can change the value of the T, D, S, H word in the buffer memory.

PART PROGRAM
The "PART PROGRAM" softkey leads to:
- EDIT
- CORRECT BLOCK
- DIRECTORY
- PROGRAM HANDLING

This key is not used to select a program for processing. After "DATA OUT", you can select program output with the softkey "PART PROGRAM".

PLANE
Select the machining plane in simulation and contour definition.

PLC ALARM
All current PLC alarms are displayed.

PLC BITS
Display or altering of the PLC bits.

PLC MESSAGE
All current PLC messages are displayed

PLC STATUS
The PLC status shows the current state of all inputs, outputs, flags, timers, counters and data words on the screen:
- E = Input word
- A = Output word
- M = Flag word
- T = Timer
- Z = Counter
- DB = Data block
- DW = Data word

PROG HANDL
(program handling)
You can copy, rename and delete the programs in the program memory.

PROGRAM CONTROL
(Influencing the program)
The key leads to the following functions:
- SKIP BLOCK
- DRY RUN FEEDRATE
- PROGRAMMED STOP
- RAPID OVERRIDE
- DECODING SINGLE BLOCK
- DRF ENABLE
- DRF HANDWHEEL

PROGRAM END
You select the functions of the "M02" (end of program) group via softkeys.

PROGR. ZO
(programmable zero offset)
Display or entering of the zero offset values for G58 and G59.

RENAME
You can change the program number.
The program itself remains unchanged.

REORG
not active

RESET
Simulation is interrupted and returned to the reset state.

ROTAT. ANGLE
(Angle of rotation)
Input of setting data for coordinate system rotation.

R PARAMETER
After "DATA OUT", you can select the output of R parameters with the "R PARAMETER" softkey.

R PARAMETERS
You input the R parameters as setting data.

SCALE MODIF.
(Scale modification)
Input of setting data for scale modification.

SELECT PROGRAM
Call-up main program or subroutine

SETTING DATA
Using setting data, the operator (user) fixes certain operating states. Setting data are adjustable for:
- Programmable and settable zero offsets
- External zero offsets
- R parameters
- Spindle data
- Axial data
- Angle of rotation
- Scale modification
- Data transfer
- General data (setting data bits).

SIMULATION
To test the program the programmed movements are shown on the SCREEN display. Programming errors are displayed as alarms.

SINGLE BLOCK
After "Program start" only one block is processed. The next blocks is only processed after another operation of "Program Start".

SKIP YES-NO
With "YES" selected, the blocks marked with an oblique (/) are skiped during program processing (Program control).

SPECIAL BLOCK
You select G04, G92, M19, G58, G50 and G51 with this softkey.

SPECIAL FUNCT.
The special functions M00, M01. (1st M Group) and M36, M37 (4th M Group) are grouped together in this function.

SPINDLE
You select the M function for the spindle motion with this softkey.
Preselect: "GUIDING" softkey

START
You activate the selected softkey function.

STOP
You stop the activated softkey function.

STORE
PRESET: not active
GUIDING: store without leaving the menu.

STORE CHOICE
Store the values entered and jump back to the selection menu

STORE MENU
Store the values entered and jump back to the main menu

STORE PLANE
Planes defined via machine data (G17, G18, G19) or by manual input (G16) are stored as the basic plane with this softkey and used for further program execution ("flexible plane selection").

SUBROUT.
(Subroutine)
This softkey is used to display:
- The stored subroutine numbers
- The number of characters used
- The free memory space

SUBROUT. START
(Subroutine Start)
Activation of output of subroutines via the universal interface.

SW VERSION
This softkey is used for display the software version.

TEACHIN PL BACK
not active

THREAD
Select the G functions for thread cutting with this softkey.

TO AUTOM
(Automatic tool offset)
By traversing to the desired reference plane, the tool offset can be measured and stored.

TOOL OFFSET
The tool offset takes into account the tool dimensions and wear. The tool offset is stored under a tool offset number, D1 to D99, in the tool offset memory.
Via the "TOOL OFFSET" softkey after "DATA OUT", you select the output of tool offsets.

UNLOCK
not active

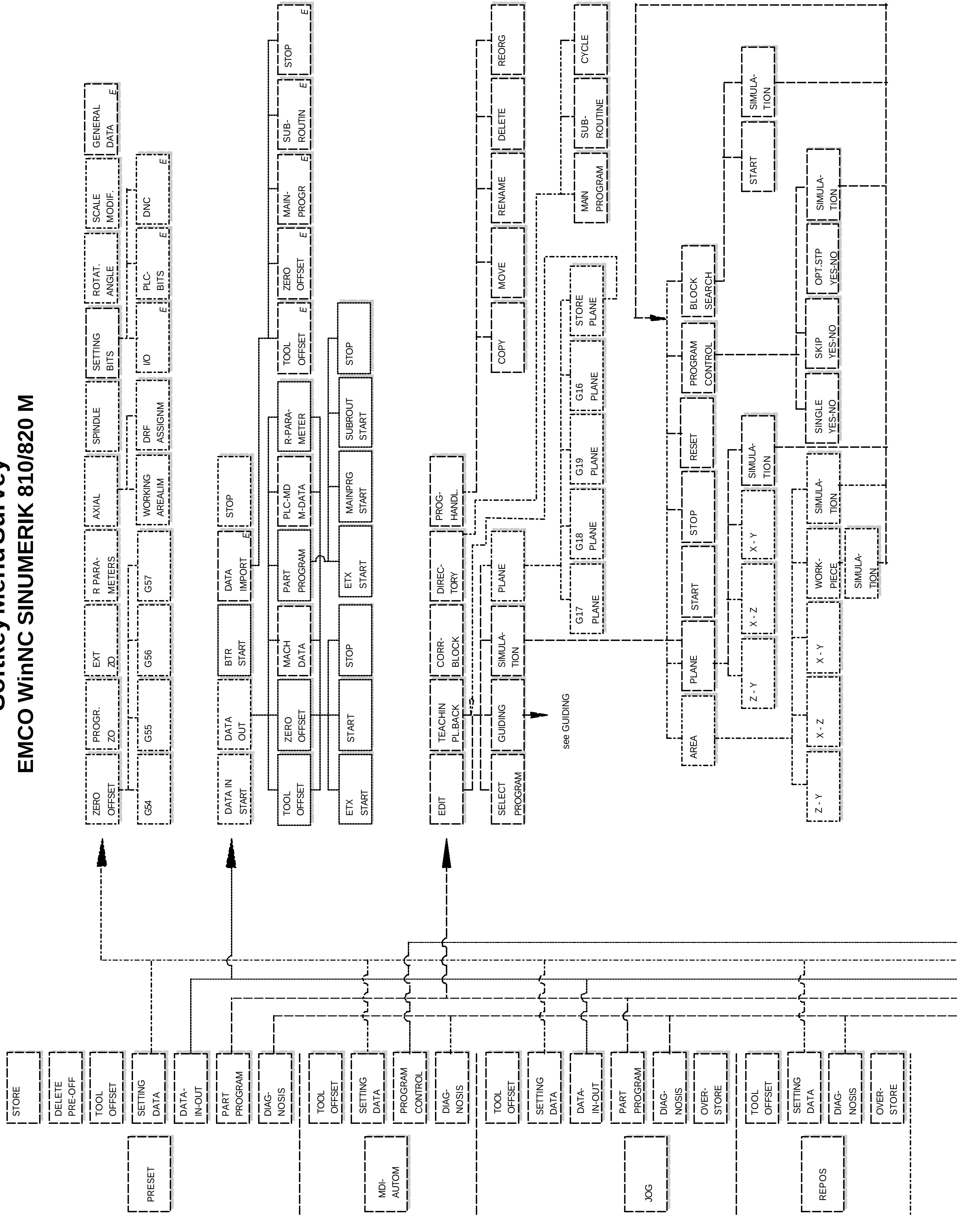
WORK CYCLE
(Machining cycle)
Using a softkey you can select cycles for frequently occurring machining sequences.
Preselect: "GUIDING" softkey.

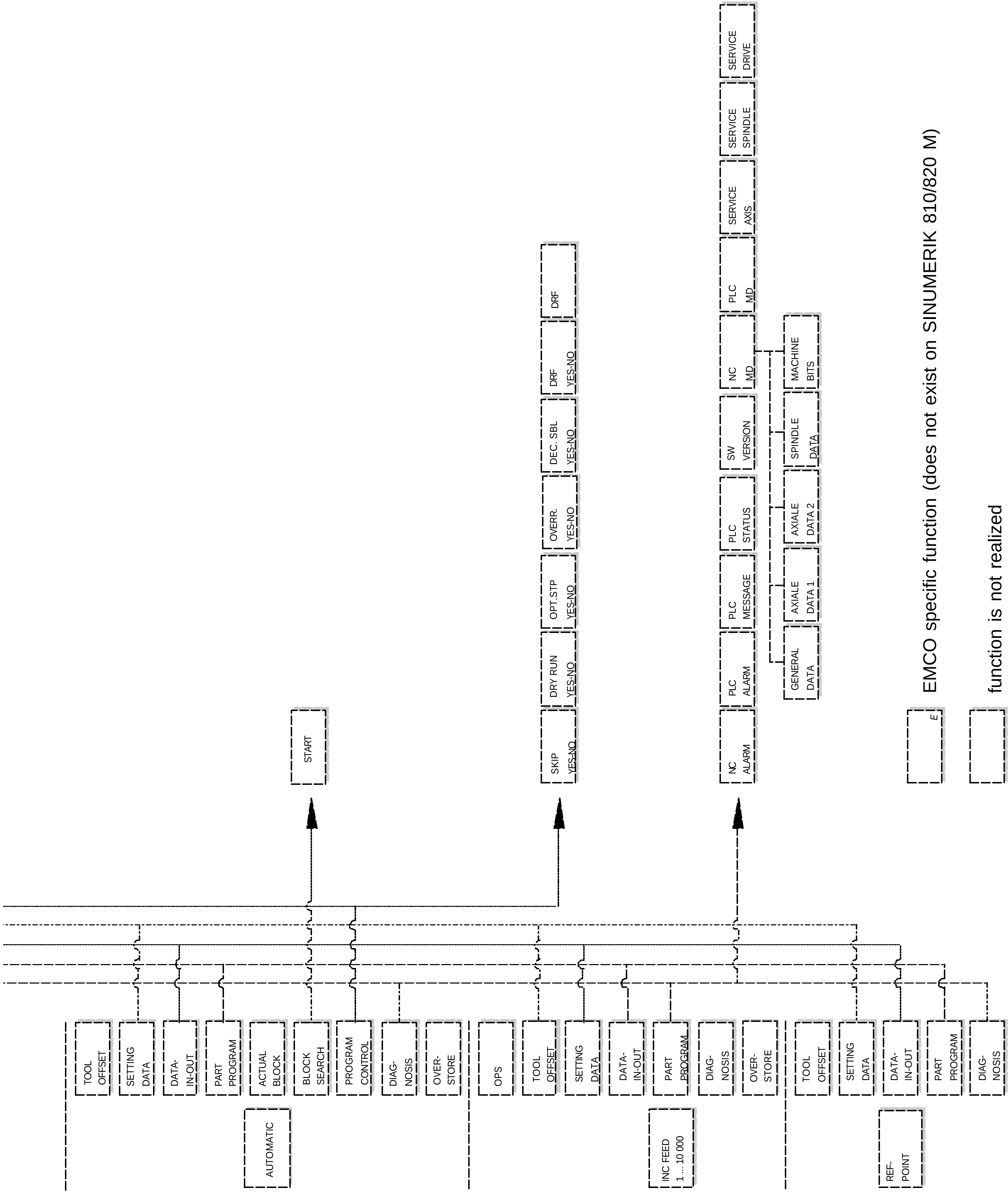
WORKING AREA LIM.
(Working area limitation)
The minimum and maximum working area limits of the defined axes appear in the display. You can modify the values displayed.

ZERO OFFSET
The settable zero offsets are put in as setting data. Programmable and external zero offsets are displayed on the SCREEN.

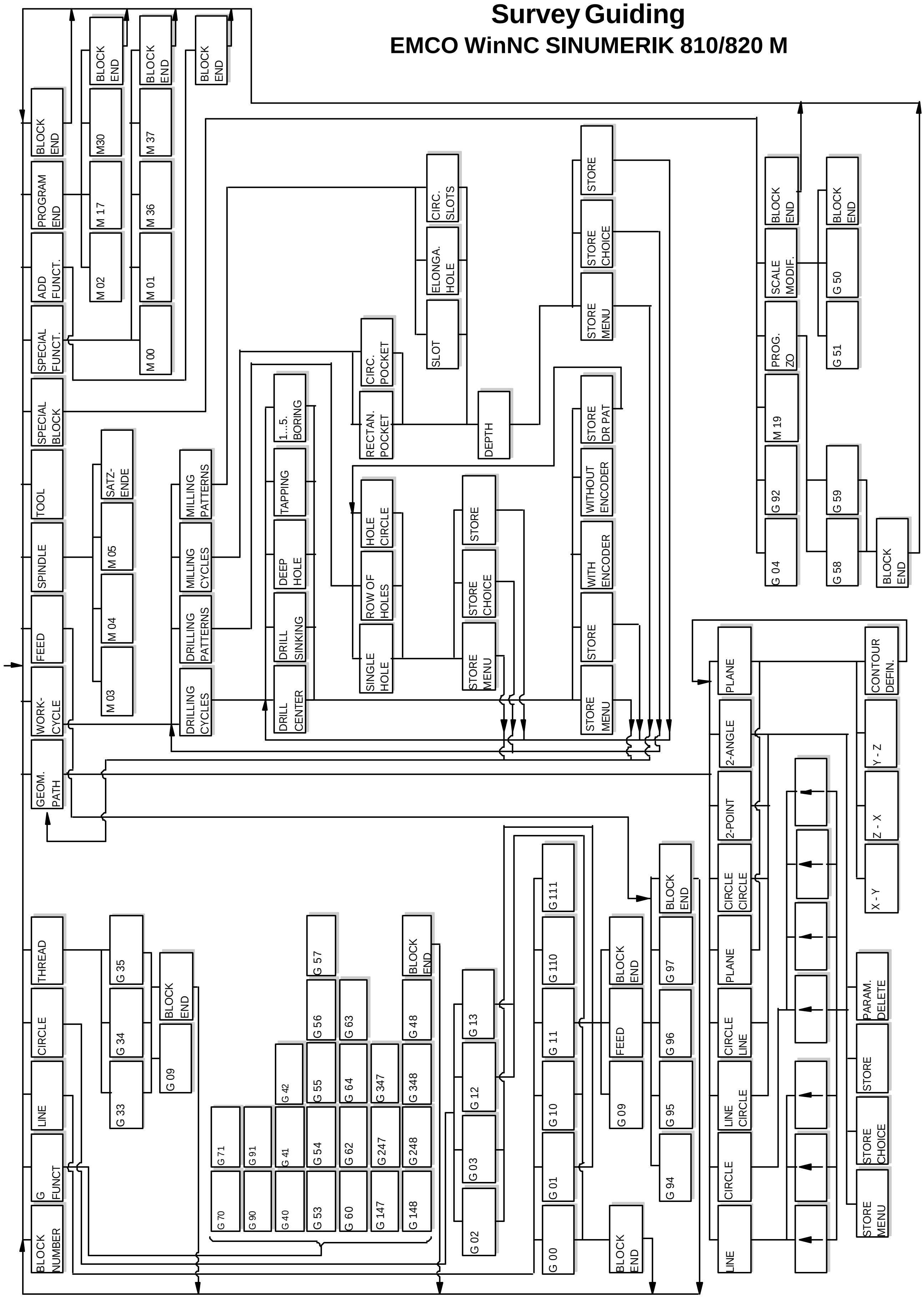
ZO AUTOM
(Automatic zero offset)
By traversing to the desired reference plane, the zero offset can be measured and stored.

Softkey Menu Survey
EMCO WinNC SINUMERIK 810/820 M




AUTOMATIC
TOOL OFFSET
SETTING DATA
DATA-IN-OUT
PART PROGRAM
ACTUAL BLOCK
BLOCK SEARCH
PROGRAM CONTROL
DIAG-NOSIS
OVER-STORE
START
SKIP YES-NO
DRY RUN YES-NO
OPT.STP YES-NO
OVERR. YES-NO
DEC. SBL YES-NO
DRF YES-NO
DRF
INC FEED 1 ... 10 000
OPS
TOOL OFFSET
SETTING DATA
DATA-IN-OUT
PART PROGRAM
DIAG-NOSIS
OVER-STORE
NC ALARM
PLC ALARM
PLC MESSAGE
PLC STATUS
SW VERSION
NC MD
PLC MD
SERVICE AXIS
SERVICE SPINDLE
SERVICE DRIVE
GENERAL DATA
AXIALE DATA 1
AXIALE DATA 2
SPINDLE DATA
MACHINE BITS
REF-POINT
TOOL OFFSET
SETTING DATA
DATA-IN-OUT
PART PROGRAM
DIAG-NOSIS
E
EMCO specific function (does not exist on SINUMERIK 810/820 M)
function is not realized

Survey Guiding
EMCO WinNC SINUMERIK 810/820 M



H: Alarms and Messages

Startup Alarms

These alarms can occur only with starting WinNC or WinCTS.

0001 Error creating the file ...
Remedy: Check whether the directories exist, which are entered in the .INI files.
Check whether writing access is possible to these directories.
Check whether the disk is full.

0002 Error opening the file ...
Remedy: Check whether the directories exist, which are entered in the .INI files.
Check whether reading access is possible to these directories (number of simultaneous open files).
Copy the correct file in the corresponding directory.

0003 Error reading the file ...
see 0002.

0004 Error writing the file ...
see 0001

0005 Too less RAM ...
Remedy: Close other WINDOWS applications
Restart WINDOWS

0006 Non compatible software version ...
Remedy: Software update.

0007 Invalid licence version ...
Remedy: Contact EMCO.

0011 Serial interface ... for digitizer is already in use
Cause: The serial interface no. ... is already occupied by another device.
Remedy: Remove the other device and connect the digitizer or define another serial interface for the digitizer.

0012 Serial interface ... for control keyboard is already in use
analogous to 0011

0013 Invalid settings for serial interface ...
Cause: The actual settings of the serial interface are not allowed for WinNC.
Allowed settings:
Baud rate: 110, 300, 600, 1200, 2400, 4800, 9600, 19200
Number of data bits: 7 or 8
Number of stop bits: 1 or 2
Parity: none, even or odd
Remedy: Change the settings of the serial interface in the WINDOWS system control (connections).

0014 Serial interface ... not present
Remedy: Select an existing serial interface.

0015- 0023 (various alarms)
Remedy: Restart WINDOWS. If these alarm occur repeatedly, contact EMCO.

0024 Invalid entry for control keyboard interface in the profile ...
Cause: The connection of the control keyboard in the file project.ini is invalid.
Remedy: Setting with WinConfig

0025 Invalid entry for digitizer interface in the profile ...
analogous to 0024

0026 Invalid entry for notebook option in the profile ...
Cause: The notebook entry in the file project.ini is invalid.
Remedy: Setting with WinConfig

0027 Error creating the start window
Remedy: Restart WINDOWS. If this alarm occurs repeatedly, contact EMCO.

0028 Invalid entry for window representation in the profile ...
Cause: The presentation entry in the file project.ini is invalid.
Remedy: Setting with WinConfig

0029 Error initializing a timer
Remedy: Close all other WINDOWS applications or restart WINDOWS.

0030 Windows 3.1 or higher required
WinNC requires WINDOWS version 3.1 or higher.

0031 - 0036 (various alarms)
see 0002

0037 Memory allocation failure
Remedy: Close all other WINDOWS applications or restart WINDOWS.

0038 Unauthorized software version
Contact EMCO.

0039 Project data non compatible to software version
Possible error after updates, contact EMCO.

0040 Invalid entry for DNC interface in the profile ...
Cause: The DNC entry in the file project.ini is invalid.
Remedy: Setting with WinConfig

0100 Mailslot could not be created
Cause: Insufficient memory in the memory area below 640 kB.
Remedy: Close all other applications, restart WINDOWS. If this is not successfull, remove not necessary device and drivers entries in config.sys or load them in the upper memory area.

0101 For WinCTS Windows for Workgroups 3.11 or higher is required
WinCTS requires WINDOWS for WORKGROUPS version 3.11 or higher.

0102 Error creating the reference table for keybitmaps
Remedy: Restart WINDOWS. If this alarm occurs repeatedly, contact EMCO.

0103 Invalid entry for WinCTS status in the profile ...
Cause: The CTS entry in the file project.ini is invalid.
Remedy: Contact EMCO

0104 Error getting the workgroup name
Remedy: Restart WINDOWS. If this alarm occurs repeatedly, contact EMCO.

0105 No workgroup found
Remedy: Insert the computer into the workgroup for WinCTS, if necessary set up a workgroup for the WinCTS computers.

0106 Invalid entry for the number of keys to record in the profile ...
Cause: The KeyFifoSize entry in the file winnc.ini is invalid.
Remedy: Correct the number, e.g. 50 (see WinConfig).

0107 - 0110 (various alarms)
Remedy: Restart WINDOWS. If this alarm occurs repeatedly, contact EMCO.

Control Alarms

These alarms can occur only with operating and programming the control functions or with running CNC programs.

16 Parity error (RS232)
Cause: data I/O: data transmition error character overflow
Remedy: set same parity for sender and receiver

17 Overflow error (RS232)
Cause: data I/O: data transmition error parity

18 Frame error (RS232)
Cause: data I/O: data transmition error data frame
Remedy: ev. set same configuration for both RS232

30 PP memory overflow (RS232)
Cause: data I/O: unknown data type, error data

32 Data format error (RS232)
Cause: data I/O: error while opening file

1040 ORDxx DAC limit X
Cause: step motor in X felt out of pace
Remedy: e.g. select lower feed or chip crosssection, check slides for smooth running, approach reference point

1041 ORDxx DAC limit Y
see alarm 1040

1042 ORDxx DAC limit Z
see alarm 1040

1480 ORDxx SW overtravel switch X
Cause: software end switch in X overtravelled
Remedy: traverse back manually

1481 ORDxx SW overtravel switch Y
see alarm 1480

1482 ORDxx SW overtravel switch Z
see alarm 1480

1560 Nxxxx Set speed to high/low
Cause: too high/low S-value
Remedy: correct program, enter belt gage in setting data

2040 Block not in memory
Cause: Blocksearch target not found
Remedy: correct program

2041 CNC program not foand
Cause: NC Program (subroutine, contour routine) not found; with NC start no program was selected
Remedy: correct call up or create program

2046 Block greater than 120 characters
Cause: NC block is too long (with loading a program)
Remedy: divide NC block in several blocks with DOS editor

2048 Nxxxx Circle end point error
Cause: invalid circle, invalid circle centre, radius too small
Remedy: correct program

2060 Nxxxx Pogramming error TO, ZO
Cause: D number too great, tool type not allowed
Remedy: correct program

2061 Nxxxx General program error
Cause: NC block structure wrong, block number, M/H/G code not allowed, invalid combination of G commands, radius/dwell programmed twice, dwell time too long, circle already programmed, general block structure error with calculating operation
Remedy: correct program

2062 Nxxxx Feed missing / not progr.
Cause: F value too great, attempt to start with F=0, also with G95/96, if S=0 or M5
Remedy: correct program

2063 Nxxxx Thread lead to high
Cause: thread pitch too great / missing, feed with threads reaches 80% of rapid speed
Remedy: correct program, smaller pitch or lower speed with threads

2065 Nxxxx Pos behind SW overtr. switch
Cause: target point programmed out of software end switch or circle path overtravels software end switch
Remedy: correct program

2068 Nxxxx Pos behind working area
Cause: target point programmed out of working area
Remedy: correct program

2072 Nxxxx Incorrect input value
Cause: chamfer/radius too large
Remedy: correct program

2073 Nxxxx No intersection point
Cause: no intersection point with contour definition arc-line
Remedy: correct program

2074 Nxxxx Wrong angle value
Cause: no contour draft is possible with that angle
Remedy: correct program

2075 Nxxxx Incorrect radius value
Cause: wrong circle centre point or radius
Remedy: correct program

2076 Nxxxx Wrong G02/G03
Cause: wrong rotational direction for tangential draft arc-line programmed
Remedy: correct program

2077 Nxxxx Incorrect block sequence
Cause: chamfer/radius: no 2. traverse block was programmed
Remedy: correct program

2081 Nxxxx CRC not allowed
Cause: thread, change of correction direction/mirror axis/tool correction not allowed with CRC, circle not in selected CRC plane, change of plane while CRC
Remedy: correct program

2082 Nxxxx CRC not determinable
Cause: plane change CRC not allowed, invalid contour element, contour violation full circle, contour element too short
Remedy: correct program

2087 Nxxxx Coordinate rotation not allowed
Cause: new coordinate rotation while CRC was active
Remedy: correct program

2160 Nxxxx Scale factor not allowed

2171 Nxxxx Approach not possible
Cause: no tool correction, tool radius = 0 (CRC)
Remedy: select tool, enter tool radius in tool data

2172 Nxxxx Retract not possible
Cause: error with selection CRC
Remedy: correct program

2501 ORDxx Synchronisation-error AC
Remedy: RESET, report to EMCO if repeatable

2502 ORDxx Synchronisation-error AC
Remedy: RESET, report to EMCO if repeatable

2503 ORDxx Synchronisation-error AC
Remedy: RESET, report to EMCO if repeatable

2504 ORDxx Insuff. RAM for interpreter
Cause: not enough RAM memory, continuation of the program not possible
Remedy: close all other WINDOWS applications, cease program, remove resident programs (from AUTOEXEC.BAT and CONFIG.SYS), restart PC

2505 ORDxx Insuff. RAM for interpreter
see 2504

2506 ORDxx Insufficient RAM
see 2504

2507 ORDxx Referencepoint not active
Remedy: approach reference point

2508 ORDxx Intenal error NC core
Remedy: RESET, report to EMCO if repeatable

2520 ORDxx RS485 device absent
Cause: With program start not all necessary RS485 participant were present or while program run a participant failed.
(AC) Axis controller
(SPS) PLC
(MT) Controlkeyboard
Remedy: switch on RS485 participant (machine, control keyboard), check cables and connectors, check terminator plug, report to EMCO if repeatable

2521 ORDxx RS485 communication error
Remedy: switch off / on the PC, report to EMCO if repeatable

2522 ORDxx RS485 communication error
switch on/off PC, report to EMCO if repeatable

2523 ORDxx INIT error on RS485 PC-board
see "Installation of the Software, Mistakes with Installation of the Software"

2524 ORDxx Gen.-Failure RS485 PC-board
switch off/on PC, report to EMCO if repeatable

2525 ORDxx Transmit error RS485
Remedy: check RS485 cables and connectors, check terminator plug, check for external electromagnetic interferences

2526 ORDxx Transmit error RS485
see 2525

2527 ORDxx Internal error AC
Remedy: switch off/on machine, report to EMCO if repeatable

2528 ORDxx Operating system error PLC
Remedy: switch off/on machine, report to EMCO if repeatable

2529 ORDxx External keyboard error
Remedy: The controlkeyboard must be switched on after the PC when it is connected to RS232, switch off/on external keyboard, report to EMCO if repeatable

2540 ORDxx Error on storing setting-data
Cause: hard disk full, wrong path determined, missing write access
Remedy: check hard disk, if repeatable reinstall software

2545 ORDxx Drive / Device not ready
Remedy: insert disk, close drive

2546 ORDxx Checksum error machine-data
Remedy: reporttoEMCOifrepeatable

2550 ORDxx PLC simulation error
Remedy: reporttoEMCOifrepeatable

2551 ORDxx PLC simulation error
Remedy: reporttoEMCOifrepeatable

2562 Read error on CNC program
Cause: error of program file, disk or hard disk error
Remedy: solve problem on DOS level, eventually reinstallsoftware

2614 Nxxxx Internal error MSD
Remedy: reporttoEMCOifrepeatable

2650 Nxxxx Wrong Cycle-Call
Cause: invalid cycle call, if cycle will be called with G code
Remedy: reporttoEMCOifrepeatable

2849 Nxxxx Internal error CRC
Remedy: reporttoEMCOifrepeatable

2904 Nxxxx Helix Z path too large
The angle between the helix and the tangent to the arc must be less than 45°.
Remedy: correct program

3000 Nxxxx General Program error
Cause: no circle parameter programmed, wrong plane for circle (G17/18/19), chamfer/ radius/thread pitch already programmed, thread pitch missing, ...
Remedy: correct program

3001 Nxxxx More than 5 geomety parameter
Cause: wrong parameter for contour draft
Remedy: correct program

3002 Nxxxx Polar/radius error
Cause: No centre point specified, centre point in wrong plane (G17/18/19)
Remedy: correct program

3003 Nxxxx Invalid address
The entered address is not allowed.
Remedy: correct program

3004 Nxxxx CL800 Error
Cause: general cycle error with cycle language CL-800, wrong parameter, invalid value, incorrect contour program, M17 in contour program missing
Remedy: correct program

3005 Nxxxx Contour definition error
Cause: no valid contour draft with the programmed coordinates, invalid parameter, invalid values
Remedy: correct program

3006 Nxxxx Wrong block structure
Cause: too much M/H/T/D/S/G commands in one block, G commands of the same group twice programmed in block, circle with more than 2 axes or polar coordinate centre point with more than 2 axes, scale factor programmed twice in a block, change of plane / zero offset / tool change / mirror while contour draft, no valid contour draft with the programmed addresses
Remedy: correct program

3008 Nxxxx Subroutine error
Cause: subroutine counter invalid, subroutine end with M30, nesting depth override
Remedy: correct subroutine counter
program M17, max. 4 subroutines nested

3011 Nxxxx To many axes/axis twice
Cause: the position (axis) was already programmed, too much axes with block with thread/ mirror
Remedy: correct program

3012 Nxxxx Block not in memory
Cause: program end without M30, jump target not found
Remedy: correct program

3021 Nxxxx CRC contour error
Cause: The tool radius is too large in relation to the smallest contour element, contour violation with full circle, programmed arc radius smaller than the tool radius
Remedy: correct program

3042 Wrong picture
Remedy: switch off / on the PC, reprot to EMCO if repeatable

3049 Wrong simulation area
Cause: With graphik simulation no or an invalid simulation area was put in
Remedy: put in / correct simulation area

3081 Nxxxx CRC not selected for approaching
Cause: With soft approaching / leaving a contour G41 or G42 must be active
Remedy: correct program

4100 Nxxxx No D-number active
Cause: cycle alarm: no tool offset active
Remedy: activate tool correction (D)

4101 Nxxxx Cutter radius = 0
Remedy: enter cutterradius

4102 Nxxxx Cutter radius too large
Remedy: correct cutterradius

4103 Nxxxx Tool to wide
Remedy: L93: use other tool

4120 Nxxxx Spindle rotat. dir. n. progr.
Remedy: L94: program spindle direction

4121 Nxxxx Spindle not in tol. range

4122 Nxxxx Calculated feed too large
Remedy: L841: reduce feed

4140 Nxxxx Fin. part diam. too small
Remedy: alter program

4180 Nxxxx Option not available !
Remedy: order option (L841, L94)

4200 Nxxxx Check definition of R
Remedy: correct R32

6000 - 7999 Machine alarms
see "Machine Alarms"

8004 ORDxx Failure main-drive unit
8005 - 8009 ORDxx Internal error AC
Remedy: report to EMCO if repeatable

8010 ORDxx Syncr. error main drive
Cause: synchronisation mark missing for main drive
Remedy: report to service technician, if repeatable

8011 - 8013 ORDxx Internal error AC
Remedy: report to EMCO, if repeatable

8014 ORDxx Decel.-time of axis too high
Remedy: report to service technician, if repeatable

8018 ORDxx Internal error AC
Remedy: report to EMCO, if repeatable

8021 ORDxx Internal error AC
Remedy: report to EMCO, if repeatable

8022 ORDxx Internal error AC
Remedy: report to EMCO, if repeatable

8023 ORDxx Invalid Z value for helix
Cause: The Z value of the helix must be smaller than the length of the arc to be traversed
Remedy: Program correction

8100 Fatal init error AC
Cause: Internal error
Remedy: Restart software or reinstall when necessary, report to EMCO, if repeatable.

8101 Fatal init error AC
see 8101.

8102 Fatal init error AC
see 8101.

8103 Fatal init error AC
see 8101.

8104 Fatal system error AC
see 8101.

8105 Fatal init error AC
see 8101.

8106 No PC-COM card found
Cause: PC-COM board can not be accessed (ev. not mounted).
Remedy: Mount board, adjust other address with jumper

8107 PC-COM card not working
see 8106.

8108 Fatal error on PC-COM card
see 8106.

8109 Fatal error on PC-COM card
see 8106.

8110 PC-COM init message missing
Cause: Internal error
Remedy: Restart software or reinstall when necessary, report to EMCO, if repeatable.

8111 Wrong configuration of PC-COM
see 8110.

8113 Invalid data (pccom.hex)
see 8110.

8114 Programming error on PC-COM
see 8110.

8115 PC-COM packet acknowledge missing
see 8110.

8116 PC-COM startup error
see 8110.

8117 Fatal init data error (pccom.hex)
see 8110.

8118 Fatal init error AC
see 8110, ev. insufficient RAM memory

8119 PC interrupt no. not valid
Cause: The PC interrupt number can not be used.
Remedy: Find out free interrupt number in the Windows95 system control (allowed: 5,7,10, 11, 12, 3, 4 und 5) and enter this number in WinConfig.

8120 PC interrupt no. unmaskable
see 8119

8121 Invalid command to PC-COM
Cause: Internal error or defective cable
Remedy: Check cables (screw it); Restart software or reinstall when necessary, report to EMCO, if repeatable.

8122 Internal AC mailbox overrun
Cause: Internal error
Remedy: Restart software or reinstall when necessary, report to EMCO, if repeatable.

8123 Open error on record file
Cause: Internal error
Remedy: Restart software or reinstall when necessary, report to EMCO, if repeatable.

8124 Write error on record file
Cause: Internal error
Remedy: Restart software or reinstall when necessary, report to EMCO, if repeatable.

8125 Invalid memory for record buffer
Cause: Insufficient RAM, record time exceeding.
Remedy: Restart software, ev. remove drivers etc. to gain more RAM, reduce record time.

8126 AC Interpolation overrun
Cause: Ev. insufficient computer performance.
Remedy: Set a longer interrupt time in WinConfig. This may result in poorer path accuracy.

8127 Insufficient memory
Cause: Insufficient RAM
Remedy: Close other programs, restart software, ev. remove drivers etc. to gain more RAM.

8128 Invalid message to AC
Cause: Internal error
Remedy: Restart software or reinstall when necessary, report to EMCO, if repeatable.

8129 Invalid MSD data - axisconfig.
see 8128.

8130 Internal init error AC
see 8128.

8130 Internal init error AC
see 8128.

8132 Axis accessed by multiple channels
see 8128.

8133 Insufficient NC block memory AC
see 8128.

8134 Too much center points programmed
see 8128.

8135 No centerpoint programmed
see 8128.

8136 Circle radius too small
see 8128.

8137 Invalid for Helix specified
Cause: Wrong axis for helix. The combination of linear and circular axes does not match.
Remedy: Program correction.

8140 Maschine (ACIF) not responding
Cause: Machine off or not connected.
Remedy: Switch on machine or connect.

8141 Internal PC-COM error
Cause: Internal error
Remedy: Restart software or reinstall when necessary, report to EMCO, if repeatable.

8142 ACIF Program error
Cause: Internal error
Remedy: Restart software or reinstall when necessary, report to EMCO, if repeatable.

8143 ACIF packet acknowledge missing
see 8142.

8144 ACIF startup error
see 8142.

8145 Fatal init data error (acif.hex)
see 8142.

8146 Multiple request for axis
see 8142.

8147 Invalid PC-COM state (DPRAM)
see 8142.

8148 Invalid PC-COM command (CNo)
see 8142.

8149 Invalid PC-COM command (Len)
see 8142.

8150 Fatal ACIF error
see 8142.

8151 AC Init Error (missing RPG file)
see 8142.

8152 AC Init Error (RPG file format)
see 8142.

8153 FPGA program timeout on ACIF
see 8142.

8154 Invalid Command to PC-COM
see 8142.

8155 Invalid FPGA packet acknowledge
see 8142 or hardware error on ACIF board (contact EMCO Service).

8156 Sync within 1.5 revol. not found
see 8142 or Bero hardware error (contact EMCO Service).

8157 Data record done
see 8142.

8158 Bero width too large (referencing)
see 8142 or Bero hardware error (contact EMCO Service).

8159 Function not implemented
Bedeutung: In normal operation this function can not be executed

8160 Axis synchronization lost axis 3..7
Cause: Axis spins or slide is locked, axis synchronisation was lost
Remedy: Approach reference point

8164 Software limit switch max axis 3..7
Cause: Axis is at traverse area end
Remedy: Retract axis

8168 Software limit overtravel axis 3..7
Cause: Axis is at traverse area end
Remedy: Retract axis

8172 Communication error to machine
Cause: Internal error
Remedy: Restart software or reinstall when necessary, report to EMCO, if repeatable. Check connection PC - machine, eventually eliminate distortion sources.

8173 INC while NC program is running

8174 INC not allowed

8175 MSD file could not be opened
Cause: Internal error
Remedy: Restart software oder bei Bedarf neu installieren, report to EMCO, if repeatable.

8176 PLS file could not be opened
see 8175.

8177 PLS file could not be accessed
see 8175.

8178 PLS file could not be written
see 8175.

8179 ACS file could not be opened
see 8175.

8180 ACS file could not be accessed
see 8175.

8181 ACS file could not be written
see 8175.

8182 Gear change not allowed

8183 Gear too high

8184 Invalid interpolaton command

8185 Forbidden MSD data change
see 8175.

8186 MSD file could not be opened
see 8175.

8187 PLC program error
see 8175.

8188 Gear command invalid
see 8175.

8189 Invalid channel assignement
see 8175.

8190 Invalid channel within message

8191 Invalid jog feed unit

8192 Invalid axis in command

8193 Fatal PLC error
see 8175.

8194 Thread without length

8195 No thread slope in leading axis
Remedy: Program thread pitch

8196 Too manny axis for thread
Remedy: Program max. 2 axes for thread.

8197 Thread not long enough
Cause: Thread length too short. With transition from one thread to the other the length of the second thread must be sufficient to produce a correct thread.
Remedy: Longer second thread or replace it by a linear interpolation (G1).

8198 Internal error (to manny threads)
see 8175.

8199 Internal error (thread state)
Cause: Internal error
Remedy: Restart software or reinstall when necessary, report to EMCO, if repeatable.

8200 Thread without spindle on
Remedy: Switch on spindle

8201 Internal thread error (IPO)
see 8199.

8201 Internal thread error (IPO)
see 8199.

8203 Fatal AC error (0-ptr IPO)
see 8199.

8204 Fatal init error: PLC/IPO running
see 8199.

8205 PLC Runtime exceeded
Cause: Insufficient computer performance

8206 Invalid PLC M-group initialisation
see 8199.

8207 Invalid PLC machine data
see 8199.

8208 Invalid application message
see 8199.

8211 Feed too high (thread)
Cause: Thread pitch too large / missing, Feed for thread reaches 80% of rapid feed
Remedy: Program correction, lower pitch or lower spindle speed for thread

9001 unknown parameter !
Cause: SPS diagnosis, unknown parameter entered

9002 Par.-number not allowed !
Cause: SPS diagnosis, unknown parameter number entered

9003 unknown display-format !
SPS-diagnosis

9004 DB not existant !
SPS-diagnosis

9005 DW not existant !
SPS-diagnosis

9006 Invalid COM-port !
SPS-diagnosis

9007 Data-transmition activ !
dataI/O

9011 No files found !
dataI/O

9014 File already exists !
dataI/O

9015 Error while opening file !
dataI/O

9016 Error while reading file !
data I/O: error while opening a file

9017 Error while writing file !
data I/O: error while writing a file

9018 Invalid COM configuration !
dataI/O

9019 No digitizer initial data foand !
Cause: a digitizer was set, but not calibrated
Remedy: calibrate digitizer (set edge points), see "External Input Devices"

9020 No valid input !
Cause: digitizer was activated on invalied field

9021 COM-port is already used !
Cause: Another device is already connected to this COM port.

9022 Digitizer not connected
Remedy: Connect digitizer, switch on ...

9023 Control keyboard not connected
Remedy: Connect control keyboard, switch on ...

9024 General RS232 communication error
Remedy: Correct settings of the serial interface.

9500 Invalid memory for program
Cause: the PC has not enough free RAM
Remedy: Close all other WINDOWS applications, eventually remove resident programs from RAM, restart PC

9501 Error while saving program
Cause: disk full?

9502 Too less memory (loading prog.)

see 9500

9508 Selected menu not found
Remedy: report to EMCO, if repeatable

9509 Too less memory for picture
Remedy: report to EMCO, if repeatable

9510 Mem.-fail. block-search buffer
Remedy: report to EMCO, if repeatable

9511 Projection error block search
Remedy: report to EMCO, if repeatable

9540 Error in BFM / BFM not foand
Remedy: report to EMCO, if repeatable

Machine Alarms

These alarms are released by the machine.
The alarms are different for the PC MILL 50/55 and the PC MILL 100/125.
The alarms 6000 - 6999 normally must be acknowledged with RESET. The alarms 7000 - 7999 are messages which will disappear usually when the releasing situation is eliminated.

PC MILL 50/55

The following alarms are valid for the PC MILL 50/55.

6000: EMERGENCY OFF
The EMERGENCY OFF key was pressed. Remove the endangering situation and restart machine and software.

6001: CYCLE TIME EXCEEDS LIMIT
Contact EMCO Service.

6002: NO PLC PROGRAM LOADED
Contact EMCO Service.

6003: DB NOT EXISTENT
Contact EMCO Service.

6004: RAM ERROR ON PLC BOARD
Contact EMCO Service.

6009: FAILURE SAFETY CIRCUIT
Defective door limit switch or main contactor. Operating the machine is not possible.
Contact EMCO Service.

6010: X-AXIS NOT READY
Step motor board defective or too hot, 24 V fuse defective. Check fuses and switch box fan filter.
Contact EMCO Service.

6011: Y-AXIS NOT READY
see alarm 6010.

6012: Z-AXIS NOT READY
see alarm 6010.

6013: MAIN DRIVE NOT READY
Main drive power supply defective, cable defective, fuse defective.
Check fuse.
Contact EMCO service.

6014: NO SPEED FOR MAIN SPINDLE
This will be released, when the spindle speed is lower than 20 rpm because of overload.
Alter cutting data (feed, infeed, spindle speed).

6019: VICE TIMEOUT
24 V fuse defective, hardware defective.
Contact EMCO service.

6020: VICE FAILURE
24 V fuse defective, hardware defective.
Contact EMCO service.

6024: DOOR NOT CLOSED
The door was opened while a machine movement. The program will be aborted.

6025: GEARBOX COVER NOT CLOSED
The gearbox cover was opened while a machine movement. A running CNC program will be aborted. Close the cover to continue.

6027: DOOR LIMIT SWITCH DEFECTIVE
The limit switch of the automatic door is displaced, defective, wrong cabled.
Contact EMCO service.

6028: DOOR TIMEOUT
The automatic door stucks, the pressured air supply is insufficient, the limit switch is displaced.
Check door, pressured air supply, limit switch or contact EMCO service.

6030: NO PART CLAMPED
No workpiece inserted, vice cheek displaced, control cam displaced, hardware defective.
Adjust or contact EMCO service.

6041: TOOL CHANGE TIMEOUT
Tool turret stucks (collision?), 24 V fuse defective, hardware defective.
A running CNC program will be stopped.
Check for a collision or contact EMCO service.

6042: TOOL CHANGE TIMEOUT
see alarm 6041.

6043: TOOL CHANGE TIMEOUT
see alarm 6041.

6044: TOOL TURRET SYNC ERROR
Hardware defective.
Contact EMCO service.

6046: TOOL TURRET SYNC MISSING
Hardware defective.
Contact EMCO service.

6048: DIVIDING TIME EXCEEDED
Dividing head stucks, insufficient pressured air supply, hardware defective.
Check for collision, check pressured air supply or contact EMCO service.

6049: INTERLOCKING TIME EXCEEDED
see alarm 6048

6050: FAILURE DIVIDING DEVICE

Hardware defective.
Contact EMCO service.

7000: INVALID TOOL NUMBER
The CNC program will be stopped.
Interrupt program with RESET and correct the program.

7007: FEED HOLD
In the robotic mode a HIGH signal is at input E3.7. Feed Stop is active until a low signal is at E3.7.

7017: GO FOR REFERENCE POINT
Approach the reference point.

7040: DOOR OPEN
The main drive can not be switched on and NC-Start can not be activated.
Some accessories can be operated only with open machine door.
Close the machine to run a program.

7043: PIECE COUNT REACHED
A predetermined number of program runs was reached. NC-Start is locked. Reset the counter to continue.

7050: NO PART CLAMPED
After switching on or after an the vice is neither at the open position nor at the closed position.
NC-Start is locked.
Traverse the vice manually on a valid end position.

7051: DIVIDING DEVICE NOT INTERLOCKED
After switching on or after an the dividing head is not in a lock position. NC-Start is locked.

PC MILL 100/125
The following alarms are valid for the PC MILL 100/125.

6000: EMERGENCY OFF
The EMERGENCY OFF key was pressed. Remove the endangering situation and restart machine and software.

6001: PLC-CYCLE TIME EXCEEDING
Contact EMCO Service.

6002: PLC - NO PROGRAM CHARGED
Contact EMCO Service.

6003: PLC - NO DATA UNIT
Contact EMCO Service.

6004: PLC - RAM MEMORY FAILURE
Contact EMCO Service.

6009: SAFETY CIRCUIT FAULT
Defective step motor system.
A running CNC program will be interrupted, the auxiliary drives will be stopped, the reference position will be lost.
Contact EMCO Service.

6010: DRIVE X-AXIS NOT READY
The step motor board is defective or too hot, a fuse is defective.
A running program will be stopped, the auxiliary drives will be switched off, the reference position will be lost.
Check fuses or contact EMCO service.

6011: DRIVE Y-AXIS NOT READY
see alarm 6010.

6012: DRIVE Z-AXIS NOT READY
see alarm 6010.

6013: MAIN DRIVE NOT READY
Main drive power supply defective, main drive too hot, fuse defective.
A running program will be stopped, the auxilliary drives will be switched off.
Check fuses or contact EMCO Service.

6014: NO MAIN SPINDLE SPEED
This will be released, when the spindle speed is lower than 20 rpm because of overload.
Alter cutting data (feed, infeed, spindle speed).

The CNC program will be aborted, the auxilliary drives will be stopped.

6024: MACHINE DOOR OPEN
The door was opened while a machine movement. The program will be aborted.

6041: TOOL CHANGE TIMEOUT
Tool drum stucks (collision?), main drive not ready, fuse defective, hardware defective.
A running CNC program will be stopped.
Check for collisions, check fuses or contact EMCO service.

6044: TOOL DISK POSITION FAULT
Position error of main drive, error of position supervising (inductive proximity switch defective or disadjusted, drum allowance), fuse defective, hardware defective.
The Z axis could have been slipped out of the toothing while the machine wasswitched off.
A running CNC program will be stopped.
Contact EMCO service.

6047: TOOL DISK UNLOCKED
Tool drum turned out of locked position, inductive proximity switch defective or disadjusted, fuse defective, hardware defective.
A running CNC program will be interrupted.
Contact EMCO service.
When the tool drum is turned out of locked position (no defect), act as following:
Turn the drum into locking position manually
Change into MANUAL (JOG) mode.
Turn the key switch. Traverse the Z slide upwards, until the alarm disappears.

6050: M25 AT RUNNING MAIN SPINDLE
Cause: Programming mistake in NC program.
A running program will be aborted.
The auxilliary drives will be switched off.
Remedy: Correct NC program

6064: DOOR AUTOMATIC NOT READY
Cause: pressure failure automatic door
automatic door stucks mechanically
limit switch for open end position defective
security print circuits defect
cabling defective
fuses defective
A running program will be aborted.
The auxilliary drives will be switched off.
Remedy: service automatic door

6072: VICE NOT READY
Attempt to start the spindle with an open vice or without clamped workpiece.
Vice stucks mechanically, insufficient compressed air supply, compressed air switch defective, fuse defective, hardware defective.
Check the fuses or contact EMCO service.

6073: DIVIDING DEVICE NOT READY
Cause: locking switch defective
cabling defective
fuses defective
A running program will be aborted.
The auxilliary drives will be switched off.
Remedy: service automatic dividing device
lock the dividing device

6074: DIVIDING TIME EXCEEDED
Cause: dividing device stucks mechanically
locking switch defective
cabling defective
fuses defective
A running program will be aborted.
The auxilliary drives will be switched off.
Remedy: service automatic dividing device

6075: M27 AT RUNNING MAIN SPINDLE
Cause: Programming mistake in NC program.
A running program will be aborted.
The auxilliary drives will be switched off.
Remedy: Correct NC program

7000: INVALID TOOL NUMBER PROGRAMMED
The tool position was programmed larger than 10.
The CNC program will be stopped.
Interrupt program with RESET and correct the program.

7016: SWITCH ON AUXILIARY DRIVES
The auxiliary drives are off. Press the AUX ON key for at least 0.5 sec. (to avoid accidentally switching on) to switch on the auxiliary drives.

7017: REFERENCE MACHINE
Approach the reference point.

When the reference point is not active, manual movements are possible only with key switch at position "setting operation".

7018: TURN KEY SWITCH
With NC-Start the key switch was in position "setting operation".
NC-Start is locked.
Turn the key switch in the position "automatic" to run a program.

7020: SPECIAL OPERATION MODE ACTIVE
Special operation mode: The machine door is opened, the auxiliary drives are switched on, the key switch is in position "setting operation" and the consent key is pressed.
Manual traversing the axes is possible with open door. Swivelling the tool turret is not possible with open door. Running a CNC program is possible only with standing spindle (DRYRUN) and SINGLE block operation.
For safety: If the consent key is pressed for more than 40 sec. the function of this key is interrupted, the consent key must be released and pressed again.

7021: INITIALIZE TOOL TURRET
The tool turret operating was interrupted.
No traversing operation is possible.
Press the tool turret key in the RESET status of the control.

7038: LUBRICATION SYSTEM FAULT
The pressure switch is defective or gagged.
NC-Start is locked. This can be reset only by switching off and on the machine.
Contact EMCO service.

7039: LUBRICATION SYSTEM FAULT
Not enough lubricant, the pressure switch is defective.
NC-Start is locked.
Check the lubricant and lubricate manually or contact EMCO service.

7040: MACHINE DOOR OPEN
The main drive can not be switched on and NC-Start cannot be activated (except special operation mode)
Close the machine to run a program.

7042: INITIALIZE MACHINE DOOR
Every movement and NC-Start are locked.
Open and close the machine door to initialize the safety circuits.

7043: PIECE COUNT REACHED
A predetermined number of program runs was reached. NC-Start is locked. Reset the counter to continue.

7054: VICE OPEN
Cause: the workpiece is not clamped
When switching on the main spindle with M3/M4 alarm 6073 (vice not ready) will be released.
Remedy: Clamp

7054: DIVIDING DEVICE NOT [illegible]
Cause: the dividing device is n[illegible]