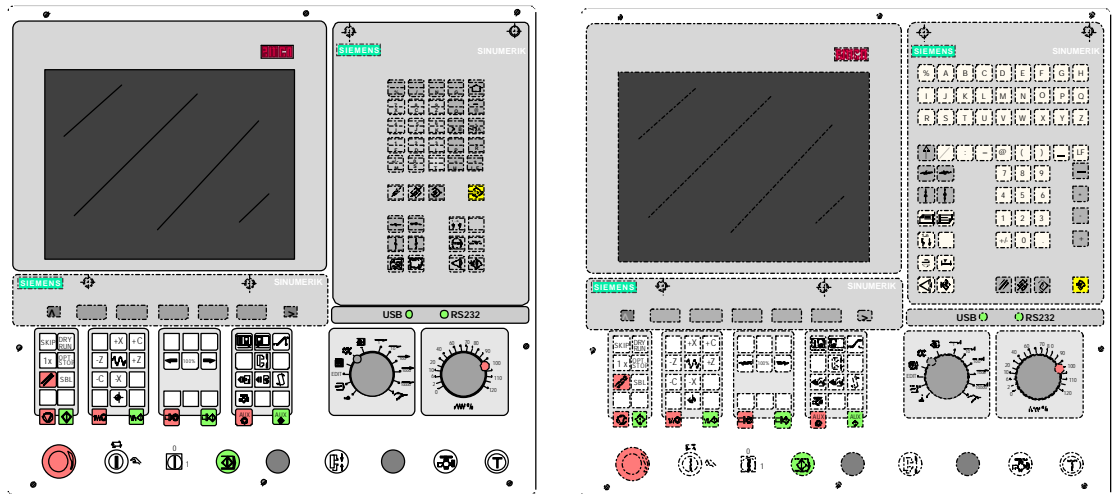# EMCO WinNC SINUMERIK 810/820 T

Software Description/ Software Version from 13.70

Software Description
EMCO WinNC SINUMERIK 810/820 T

Ref.No. EN 1804   Edition I2003-10

emco

innovative machine tools
industrial training systems

Preface

The software EMCO WinNC SINUMERIK 810/820 T Turning is a part of the EMCO education concept on PC basis.

Target of this concept is learning to operate and program the original control at the PC.

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

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 SINUMERIK 810/820 T Turning, 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 Technical Documentation
A-5400 Hallein, Austria

EMCO

Contents

EMCO

E: @-Codes

G: Survey Pages

H: Alarms and Messages

Starting Information
see attachment

emco

A: Key Description

Control Keyboard, Digitizer Overlay
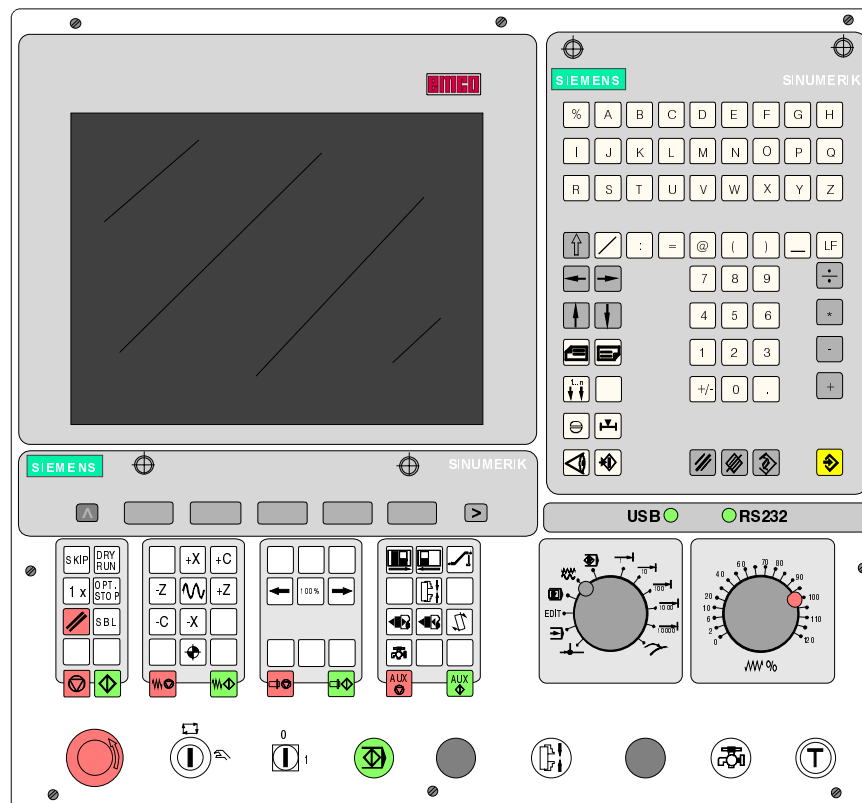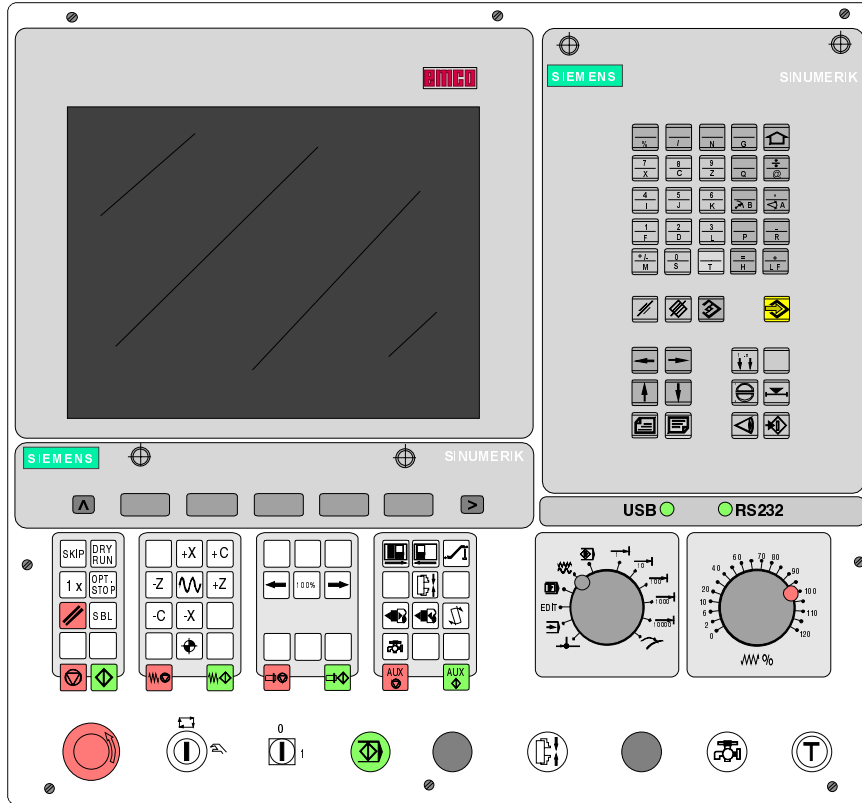
Key functions

⬗	Input key
⫽	Delete input / operator message
⬦	Delete word / block
⬦	Alter word
⬖	Search address / block / word
↑ ↓	Cursor down / up
← →	Cursor left / right
⬁ ⬂	Page up / down
⊖	Acknowledge alarm
⊢▼⊣	Actual position in double size letters

Address and Numeric Keyboard

SINUMERIK 810T:
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 pressing an address key the SHIFT function is active automatically.

6 / **K**	function K
⌂ + **6** / **K**	funkcion 6
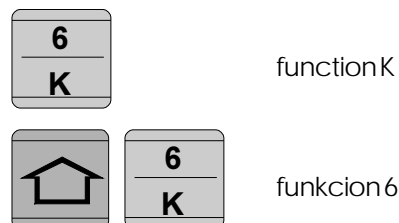
SINUMERIK 820T:
Every address or number has its own key.

Address and numeric keyboard (SINUMERIK 810 T)

%	/	N	G	⌂
7 / X	8 / C	9 / Z	Q	÷ / @
4 / I	5 / J	6 / K	⊅ B	* / ◁ A
1 / F	2 / D	3 / L	P	- / R
+/- / M	0 / S	. / T	= / H	+ / L F

Address and numeric keyboard of the SINUMERIK 810 T

A 2

A 3

Machine Control Keys

The machine control keys are in the lower block of the control keyboard resp. the digitizer overlay.
Depending on the used machine and the used accessories not all functions may be active.
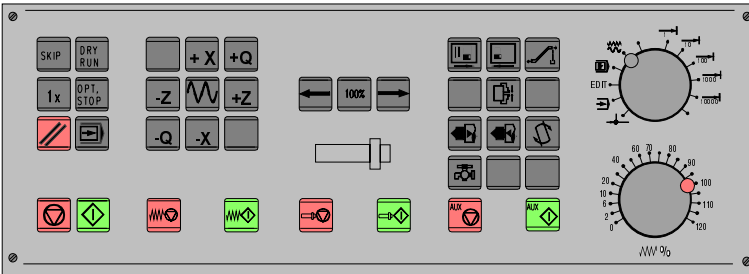
Machine control keyboard of the EMCO control keyboard
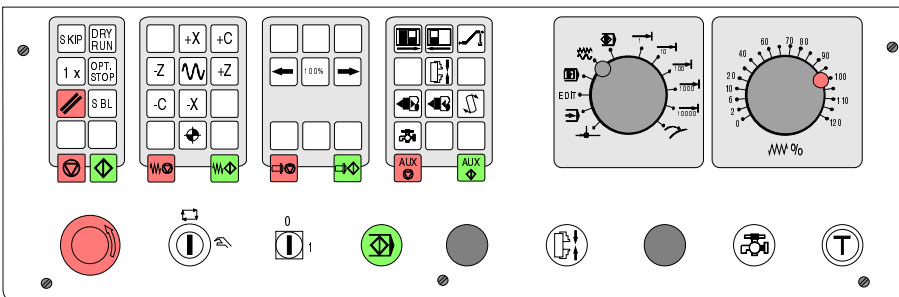
Machine control keyboard of the EMCO PC-Turn Series

SKIP	SKIP (skip blocks will not be executed)
DRY RUN	DRY RUN (test run of programs)
OPT. STOP	OPT STOP (program stop at M01)
(RESET)	R E S E T
	Single block machining
	Program stop / program start
+X +Q -Z +Z -Q -X	Manual axis movement
	Approaching the reference point in all axes
	Feed stop / feed start
← 100% →	Spindle override lower / 100% / higher
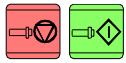
Spindel stop / spindle start; spindle start in JOG and INC1...INC10000 mode:

Clockwise: perss [key] key short, Counterclockwise: press [key] min. 1 sec.
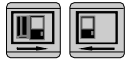
Open / close door

Close / open clamping device

Tailstock back / forward

Swivel tool holder

Coolant / puff blowing on / off
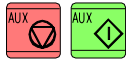
AUX OFF / AUX ON (auxiliary drives off / on)
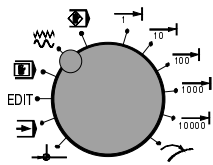
Mode selector
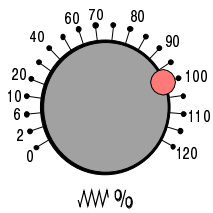
Feed / rapid feed override switch
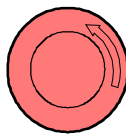
EMERGENCY OFF (Unlock: pull out button)
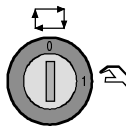
Key switch for special operations (siehe Maschinenbeschreibung)

Additional NC start key

Additional key clamping device

Consent key

No function

emco
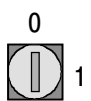
PC Keyboard

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 TURN 50/55: Puff blowing ON/OFF
EMCO PC TURN 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.

EMCO
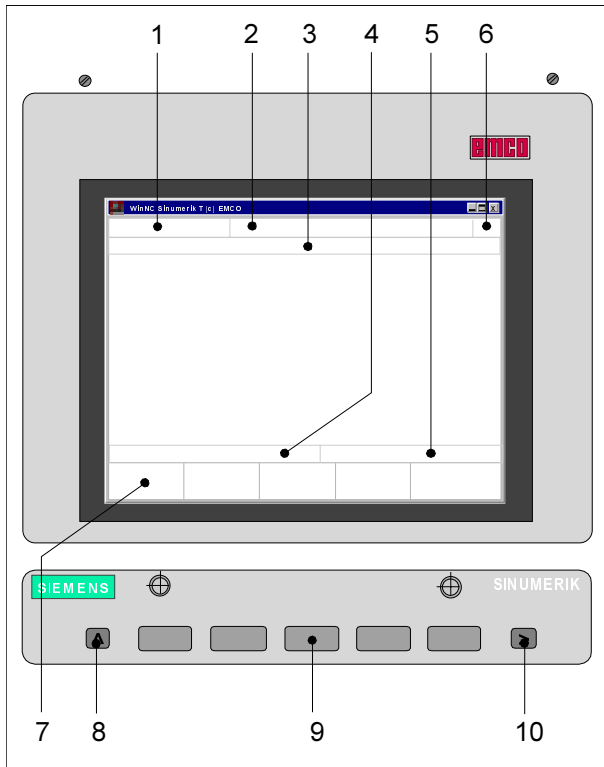
Screen with Softkeys

At the operating field the following parts are defined:

1   Display of the mode
2   Display of the operating conditions
3   Display of the alarm number, text (comment)
4   Display of the 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.

Screen with Softkeys

A 8

B: Basics

Reference Points of the EMCO Lathes

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.

Reference points in the working area

Zero Offset

With EMCO lathes the machine zero "M" lies on the rotating axis and on the end face of the spindle flange. 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.

Zero offset from the machine zero point to the workpiece zero point

Coordinate System

The X coordinate lies in the direction of the cross slide, the Z coordinate lies in the direction of the longitudinal slide.
Coordinate values in minus direction describe movements of the tool system towards the workpiece, values in plus direction away from the workpiece.

Coordinate System with Absolute Programming
The origin of the coordinate system lies at the machine zero "M" or at the workpiece zero "W" following a programmed zero offset.
All target points are described from the origin of the coordinate system by the indication of the respective X and Z distances.
X distances are indicated as the diameter (as dimensioned on the drawing)

Coordinate System with Incremental Programming
The origin of the coordinate system lies at the tool mount reference point "N" or at the cutting tip after a tool call-up.
With incremental programming the actual pathes of the tool (from point to point) are described.
X will be entered as radius.

Absolute coordinates relate to a fixed position, incremental coordinates to the tool position.
The X and -X in brackets are valid for the PC TURN 50/55 because the tool is in front of the turning centre at this machine.

```
┌─────────────────────────────────────────────────┐
│ ▣  WinNC Sinumerik T (c) EMCO            ▢□☒      │
├─────────────────────────────────────────────────┤
│ AUTOMATIC                                         │
│                                                   │
│ SETTINGDATA  ZERO OFFSET          (G54)           │
│                                                   │
│    ZERO OFFSET            ZERO OFFSET ADDITIVE     │
│      X        0              X        0            │
│      Y        0              Y        0            │
│                                                   │
│                                                   │
│                                                   │
│                                                   │
│                                                   │
│  G54  F3   G55  F4   G56  F5   G57  F6      F7    │
└─────────────────────────────────────────────────┘
```
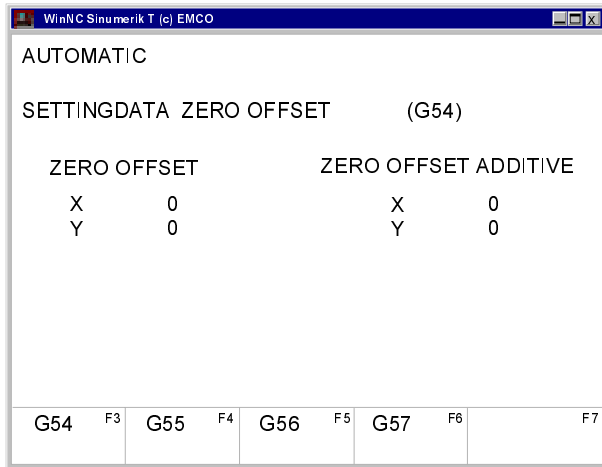
Input pattern for Zero Offset G54

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 the zero offset G54. The particular offsets G54 - G57 can be selected with softkeys.

• Below ZERO OFFSET the measured values (e.g.: X=0, Z=length of chuck) are entered.

• Corrections to this value can be entered below ZO ADDIT. These corrections will be added.

• 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.

```
┌─────────────────────────────────────────────────┐
│ ▣  WinNC Sinumerik T (c) EMCO            ▢□☒      │
├─────────────────────────────────────────────────┤
│ JOG                                               │
│ SETTINGDATA COORDINATE ROTATION                   │
│                                                   │
│       A     0        (G54)                        │
│       A     0        (G55)                        │
│       A     0        (G56)                        │
│       A     0        (G57)                        │
│                                                   │
│       A     0        (G58)                        │
│       A     0        (G59)                        │
│                                                   │
│                                                   │
│   F3      F4      F5      F6          F7          │
└─────────────────────────────────────────────────┘
```

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 at the call-up of the zero offset.

• Press the softkey SETTING DATA in any mode.

• Extend softkey line (key [>]) and press softkey ROTAT. ANGLE.

• The screen shows the input pattern for coordinate rotation. The single rotations for G54 - G57 can be entered in this pattern, the rotations for G58 and G59 are 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.

EMCO

Tool Data Measuring

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

Every tool which is used for machining has to be measured. The distances in both axis directions between tool mount reference point and tool tip are 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.

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

Tool call in program: **T4 D41**

The address T marks the position in the tool turret, the address D marks the correction number belonging to the tool.

The length corrections can be measured half-automatically, cutter radius and cutter position have to be inserted manually.

The cutter position must always be inserted!
Inserting the cutter radius is **only** necessary for using the **cutter radius compensation** with this tool.

The tool data measuring occurs for type 1-9:
L1:   in-X direction absolute from point "N" in radius
L2:   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)

Direction of the length correction for the tool types

Tool radius R

emco
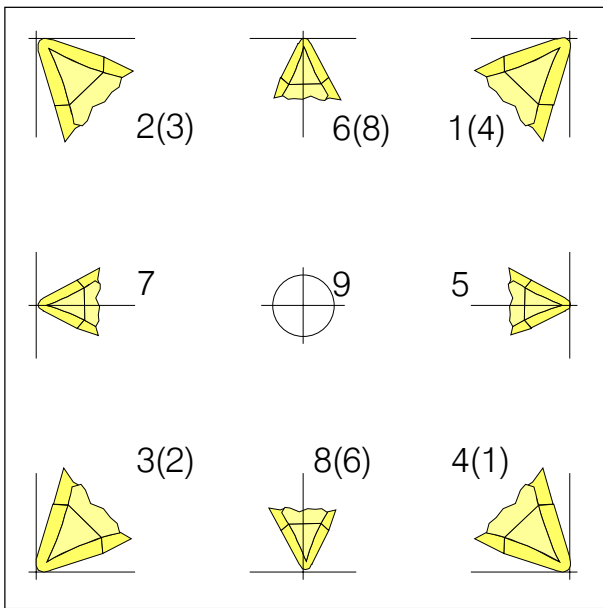
Cutter position

Cutter position (Type)

Look at the tool like it is clamped at the machine to determine the tool type.
For machines with the tool below (in front of) the turning centre (e.g. PC TURN 50/55) the values in brackets must be used because of the opposite +X direction of these machines.

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

X+/- ..... incremental in diameter
Z+/- ...... incremental
R+/- ..... incremental

emco

Input of the Tool Data

```
WinNC Sinumerik T (c) EMCO                          _ □ X

AUTOMATIC

TOOL OFFSET
Actual tool offset no.                    D
D1      0  Tool number                    0
        1  Type                           0
        2  L1 Geometry                    0.
        3  L2 Geometry                    0.
        4  Diameter/Radius                0.
        5  L1 Wear                        0.
        6  L2 Wear                        0.
        7  Diameter/Radius                0.
        8  L1 Base                        0.
        9  L2 Base                        0.


       F3          F4          F5          F6          F7
```
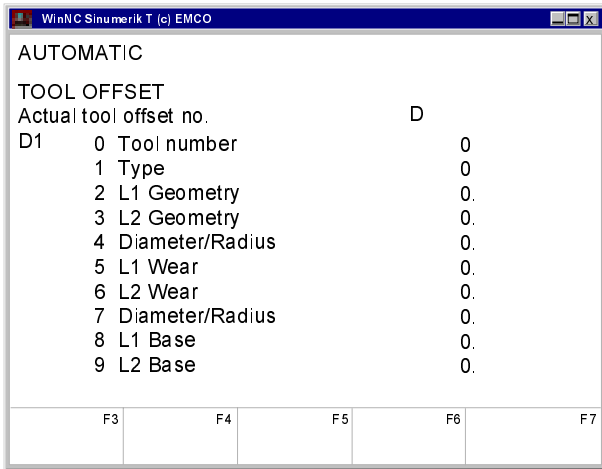
Input pattern for tool data

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 (inverse 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 ⌷ .

EMCO

Tool Data Measuring by Scraping

Manual Calculation

- Clamp a workpiece with exact measured diameter.

- Select the mode JOG

- Traverse with the tool turret disk onto the workpiece (standing spindle)
Reduce feed to 1 %
Hold a sheet of paper between workpiece and tool turret disk and traverse with the tool turret disk (tool mount reference point) so far onto the workpiece until the paper stucks.

- Read and note the actual Z position (Z1) of the slide.

- Move away the tool turret from the workpiece and swivel in the first tool to be measured.

- Traverse with the tool tip onto the face of the workpiece, insert paper, reduce feed.

- The difference between old and new Z position is the tool offset L2 = Z1 - Z.

- Traverse with the tool tip onto the circumference of the workpiece, insert paper, reduce feed.

- Read and note actual x position (X1) of the slide.

- Slide position X1 and diameter of the workpiece D are diameters; the tool offset L1 is a radius.

- $L1 = \dfrac{X1 - D}{2}$

- Enter the values L1 and L2 and the type (cutter position) and the cutter radius for the respective tool offset..

- Repeat this sequence for the following tools.

Automatic Calculation

- Clamp a workpiece with exact measured diameter.

- Select the mode JOG

- Traverse with the tool turret disk onto the workpiece (standing spindle)
Reduce feed to 1 %
Hold a sheet of paper between workpiece and tool turret disk and traverse with the tool turret disk (tool mount reference point) so far onto the workpiece until the paper stucks.

- Read the actual Z position (Z1) of the slide and key in this value into the input line.

- Press softkey TO AUTOM.

- Move cursor to the input field "reference Z" and take over the actual Z position which was put in into the input line by pressing ⬖ .

- Enter workpiece radius as "reference X".

- Move away the tool turret from the workpiece and swivel in the first tool to be measured.

- Traverse with the tool tip onto the face of the workpiece, insert paper, reduce feed.

- Enter the type (cutter position) and the cutter radius for the respective tool offset.

- Press softkey TO AUTOM.

- Enter tool offset number, ZO no. = 53 and press the softkey STORE Z-AXIS.

- Traverse with the tool tip onto the circumference of the workpiece, insert paper, reduce feed.

- Press softkey STORE X-AXIS.

- Repeat this sequence for the following tools.

Note

With drilling tools (type 10) L1 is the length of the tool in Z direction.

EMCO

Tool Data Measuring with the Optical Presetting Device

Tool data measuring with the optical presetting device occurs in principle in the same way like with scraping. The optical way is more precise because touching will be avoided and the tool is displayed enlarged in the optics.
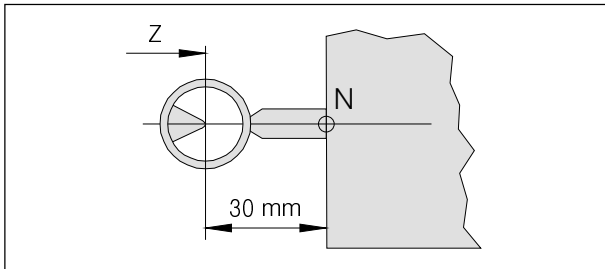
Reference tool measuring Concept Turn 50/55

Reference tool measuring Concept Turn 105/155
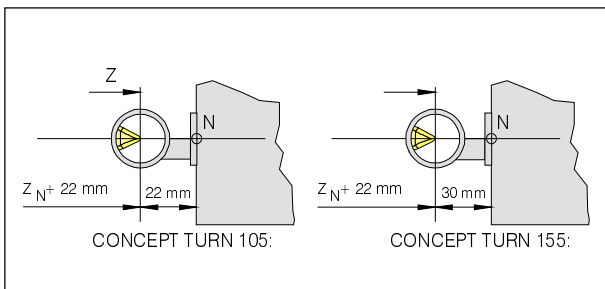
CONCEPT Turn 50/55   CONCEPT Turn 105/155
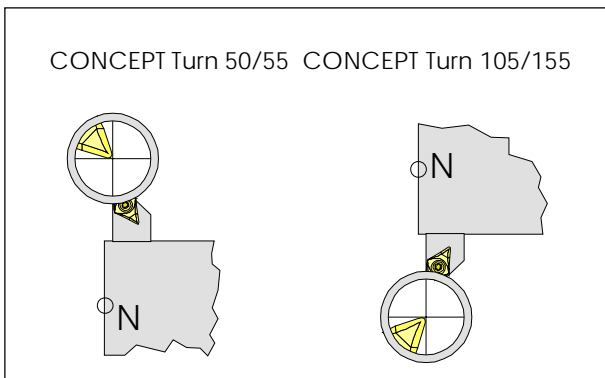
Traverse into the graticule with the tool

Manual Calculation

• Mount the optical presetting device in the working area in a way, that the measuring point can be reached with the reference tool and with all tools to be measured.
• Select the mode JOG
• Mount the reference tool at station 1 of the tool turret
• Swivel in station 1
• Traverse the tip of the reference tool into the reticule of the optics.
  Note: An object viewed through the optics is mirrored in the X and Z axis.
• Read and note actual slide position displayed at the screen..
• The tip of the reference tool is at X height of the tool mount reference point N and in Z direction 30 mm (PC TURN 50/55/155) or 22 mm (PC TURN 120/125: 20 mm tool length + 2mm outstanding length of the holder) in front of N.
  $Z_N = Z-30$ or $Z-22$ $X_N = X$
• Swivel tool turret and traverse with the first tool to be measured into the reticule.
• The difference between the old position of the tool mount reference point (X_N, Z_N) and the new position are the tool offsets L1, L2.
  The X values are in diameter and must be halved because L1 is a radius value.

• Clamp next tool etc.

Automatic Calculation

• Traverse with the reference tool into the reticule.
• The tip of the reference tool is in X at the position of the tool mount reference point N, in Z 30 mm (PC TURN 50) or 22 mm (PC TURN 120: 20 mm tool length + 2mm outstanding length of the holder) in front of N.
  $Z_N = Z-30$ or $Z-20$ $X_N = X$
• Store the coordinates X_N and Z_N in a zero offset (G54-G57).
• Swivel in the first tool.

• Traverse into the reticule with the first tool.
• Enter the type (cutter position) and the cutter radius for the respective tool offset.
• Select Softkey TO AUTOM.
• Enter tool offset number, ZO no. = (54-57) (not 53) and press the softkeys STORE X-AXIS and STORE Z-AXIS.
• Traverse the next tool into the reticule etc.

Note

With drilling tools (type 10) L1 is the length of the tool in Z direction.

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 turret.

MDI-AUTOMATIC

You can enter blocks of a part program in the intermediate store.
The control works off the inserted blocks 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.

I 1 ... I 10 000

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 $\boxed{-X}$ $\boxed{+X}$ $\boxed{-Z}$ $\boxed{+Z}$.
The selected increment (1, 10, 100, ...) must be larger than the machine resolution (smallest possible traverse path), otherwise no movement will occur.

EMCO

Approach the Reference Point

By approaching the reference point the control will be synchronized with the machine.
• Select the mode REFPOINT.

• Press the JOG keys -X or + X resp. -Z or +Z to approach the reference point in the respective direction.

• With the key REF ALL both axes will be approached automatically (PC keyboard).

Danger of Collisions

Take care of obstacles in the working area (chucks, 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 the softkey SETTING DATA.

• Extend the softkey line with the key > and press the softkey GENERAL DATA

• In the input pattern you can select the language and the workpiece directory.

Workpiece Directory

In the workpiece directory the CNC programs created by the operator will be stored.
The workpiece directory is a subdirectory of that 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 activated with restart of the software.
Input with PC keyboard:
• DT for German
• EN for English
• FR for French
• SP for Spanish
• NL for Netherlands

EMCO

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:

| N | 5 | ⬙ | Block number (not necessary) |

| G | 1 | ⬙ | 1. word |

| X | N | 3 | ⬙ | 2. word |

... ...

| LF | ⬙ | LineFeed - block end (with PC keyboard ↵) |

or ⬙
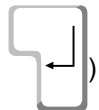
Insert Block
Position the cursor before the block, that should follow the inserted block and enter the block to be inserted.

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

emco

Program Input with Guiding Function

In an opened program you can enter blocks guided by menus.
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 CONTOUR

- 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, SELECT MEMORY or MEMORY MENU when input is finished.

- Now the contour draft will be stored as block in the part program with all the entered geometrical 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

Pricipially NC programs from CAD/CAM systems can be read in into EMCO WinNC SINUMERIK 810/820 T.

Act as following:

- The NC program must be put out in the format of the SINUMERIK 810/820.

- Rename the file.
  The NC programs of WinNC will be stored in the following way:
  %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)

EMCO

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.

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)

```
WinNC Sinumerik T (c) EMCO                          _ □ x

AUTOMATIC
DATA INPUT

Available memory:  62589 Zeichen

Data type:

Interface no. for data in:        2

Interface allocation:          1 = RTS-LINE
                               2 = RTS-LINE



DATA-IN F3  DATA-    F4   BTR     F5  DATA    F6   STOP  F7
START       OUT          START        IMPORT
```

Input pattern for data input-output

emco

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.
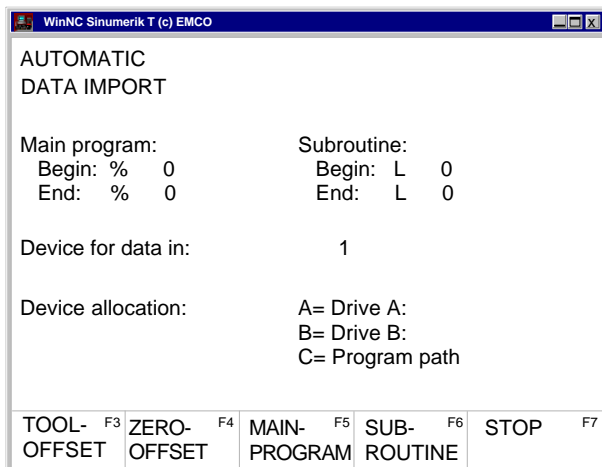
Data Import

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 MAINPROGRAM or SUBROUT INE 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 import

emco

```
WinNC Sinumerik T (c) EMCO                    _ □ X

AUTOMATIC
DATA OUTPUT

Available memory:       63548 Zeichen



Interface no. for data out:        2

Interface allocation:           1 = RTS-LINE
                                2 = RTS-LINE




TOOL-   F3  ZERO-   F4  MACH.   F5  PART-    F6  PLC-      F7
OFFSET      OFFSET      DATA        PROGRAM      M-DATA
```

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.

```
WinNC Sinumerik T (c) EMCO                    _ □ X

AUTOMATIC
DATA OUTPUT


Main program:              Subroutine:
  Begin: %   0               Begin: L   0
  End:   %   0               End:   L   0

Interface no. for data out:            2

Interface allocation:              1=RTS-LINE
                                   2=RTS-LINE




    F3  EXT    F4  MAINPRG F5  SUBROUT F6  STOP      F7
        START      START       START
```

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. SUBROUT 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.

```
WinNC Sinumerik T (c) EMCO                    _ □ X

AUTOMATIC
DATA OUTPUT ZERO OFFSET/ROTATION

Channel no.0

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

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

```
WinNC Sinumerik T (c) EMCO                    _ □ X

AUTOMATIC
DATA OUTPUT


   Available memory:   36987

   Interface no. for data out:      1
                ...
```

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

EMCO

```
┌──────────────────────────────────────────────┐
│ ▣ WinNC Sinumerik T (c) EMCO          _□x │
│                                                │
│ AUTOMATIC                                      │
│ PARAMETERS OF SERIAL INTERFACES                │
│                                                │
│                         COM1      COM2         │
│ Baudrate                9600      9600         │
│ Parity                  E         E            │
│ Stopbits                1         1            │
│ Databits                7         7            │
│                                                │
│                                                │
│ Control parameter       00000000              │
│ ETX-Character code      00000011              │
│                                                │
│                                                │
│      F3      F4        F5       F6       F7    │
└──────────────────────────────────────────────┘
```
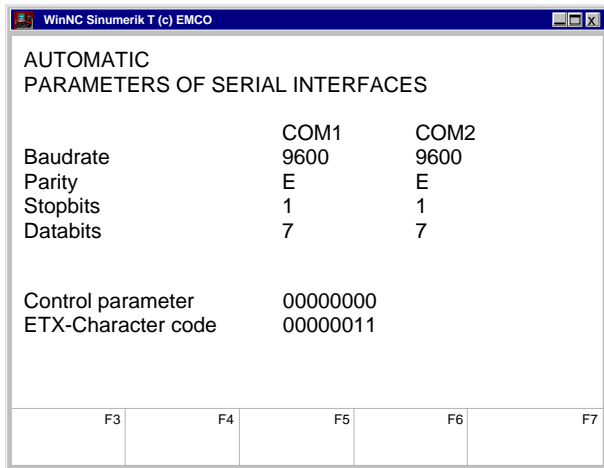
Input pattern for adjusting the serial interface

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 Transmit) 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).

emco

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.: %79: `%` `⇪` `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 certain operations (e.g. tool change).

Program Influence

By actuating the following softkeys running 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 speed, the spindle stands still. The dryrun-feed is displayed 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: for 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 the control system and the eventually connected RS 485 devices will be displayed.

Graphic simulation

NC programs can be simulated graphically.

Graphic screen

- Press the softkeys PART PROGRAM and EDIT.

- When no program is selected, enter the program number and press the softkey SELECT PRO-GRAM.

- 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.
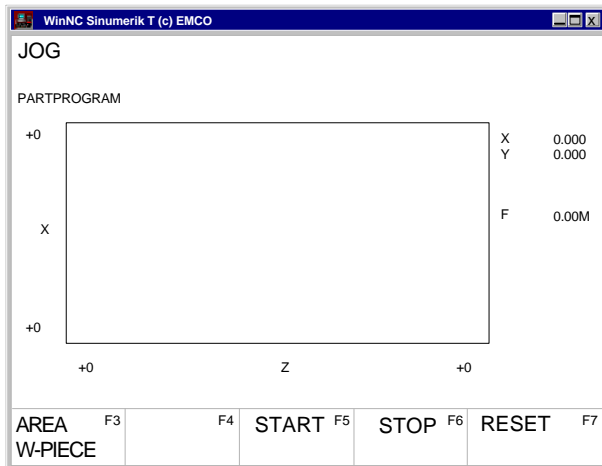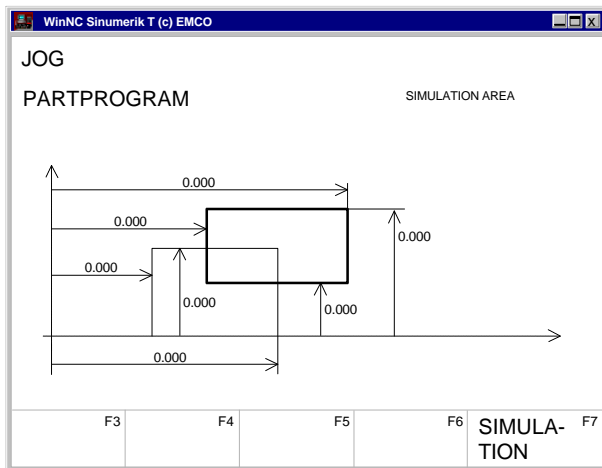
Define workpiece size and simulation area
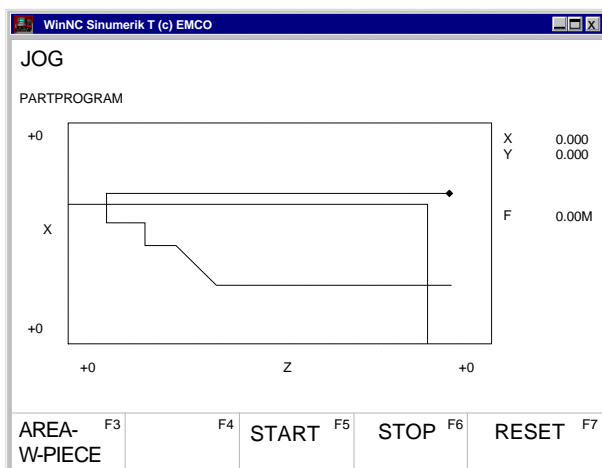
- Press the softkey AREA W-PIECE, to define the workpiece size and the simulation area.

- The fine lined rectangular shows the workpiece, the bold lined rectangular the visible screen window. The dialog line shows the meaning of the marked dimension.

- Enter the workpiece size and the simulation area.

With the softkey START the graphic simulation starts. With the softkey STOP the graphic simulation stops. With the softkey 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.

EMCO

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

- Address

- Number combinations (partly with sign)

```
%1234

N0100   G01   X15        Z-17

Z-17

Z

-17
```

Parts of an NC program

Addresses

% .... program number 1 - 9999
L .... subroutine number 1 - 9999
N .... block number 1 - 9999

G .... path function
M .... miscellaneous function

A .... angle
B .... radius (pos. sign), chamfer (neg. sign), circle radius
D .... tool offset 1 - 49
F .... feed, dwell
I, K .... circle parameter, thread pitch
P .... number of subroutine runs, scale factor
R .... parameter for cycles
S .... spindle speed, cutting speed
T .... tool call (tool turret position)
X, Z .... position data (X also dwell)
LF .... block end

EMCO

Survey of G Commands

G00 .... Positioning (rapid traverse)
G01[1] .... Linear interpolation
G02 .... Circular interpolation clockwise
G03 .... Circular interpolation counterclockwise
G04[2] .... Dwell
G09[2] .... Exact stop
G10 .... Polar coordinate interpolation, rapid traverse
G11 .... Polar coordinate interpolation, linear interpolation
G12 .... Polar coordinate interpolation, circular interpolation clockwise
G13 .... Polar coordinate interpolation, circular interpolation counterclockwise
G16 .... Plane selection with free axis selection
G18 .... Plane ZX
G25 .... Minimum working area limitation
G26 .... Maximum working area limitation
G33 .... Thread cutting
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 .... Programmable zero offset 1
G59 .... Programmable zero offset 2
G60 .... Exact stop mode
G62 .... Deselection exact stop mode
G63 .... Thread tapping with compenstion chuck
G64[1] .... Deselection exact stop mode
G70 .... Measuring in inches
G71 .... Measuring in millimetre
G80 .... Deselection of drilling cycles
G90[1] .... Absolute programming
G91 .... Incremental programming
G92[2] .... Spindle speed limit
G92 P .... Cylindrical interpolation ON
G92 P1 .... Cylindrical interpolation OFF
G94 .... Feed per minute
G95[1] .... Feed per revolution
G96 .... Constant cutting speed, feed per revolution
G97[2] .... Constant spindle speed
G130 .... Transmit OFF
G131 .... Transmit ON
G147[2] .... Soft approach to contour with linear
G148[2] .... Soft leaving with linear
G247[2] .... Soft approach to contour with quarter circle
G248[2] .... Soft leaving with quarter circle
G347[2] .... Soft approach to contour with semicircle
G348[2] .... Soft leaving with semicircle

Survey of M Commands

M00 .... Programmed stop unconditional
M01 .... Programmed stop conditional
M02 .... Main program end
M03 ...... Spindle ON clockwise
M04 ...... Spindle ON counterclockwise
M05[1] .... Spindle OFF
M08 .... Coolant ON
M09[1] .... Coolant OFF
M17 .... Subroutine end
M20 .... Tailstock BACK
M21 .... Tailstock FORWARD
M25 .... Open clamping device
M26 .... Close clamping device
M30 .... Main program end
M52 .... C- axis ON
M53 .... C- axis OFF
M71 .... Puff blowing ON
M72[1] .... Puff blowing OFF

Survey of Cycles

L93 .... Cut-in cycle
L94 .... Undercut cycle
L95 .... Stock removal with back pockets
L96 .... Stock removal without back pockets
L97 .... Thread cutting cycle
L98 .... Deep hole drilling cycle
L99 .... Thread draft
L971 ...... Longitudinal thread

[1] .... Initial status
[2] .... Effective blockwise

EMCO

Description of G Commands

G00   Positioning (Rapid Traverse)

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

The slides are traversed with 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 Z56

incremental G91
N50 G00 X-30 Z-30.5
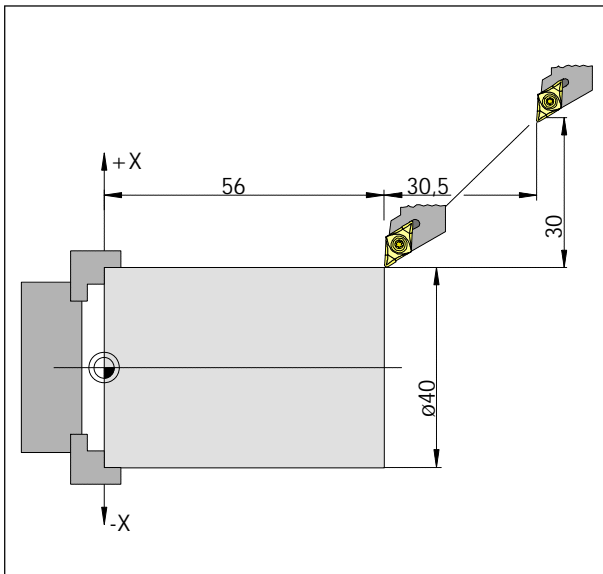
Absolute and incremental measures

G01   Linear Interpolation

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

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

Example

absolute G90
.....
N20 G01 X40 Z20.1 F0.1
or
N20 G01 X40 A158.888 F0.1

incremental G91
.....
N20 G01 X10 Z-25.9 F0.1
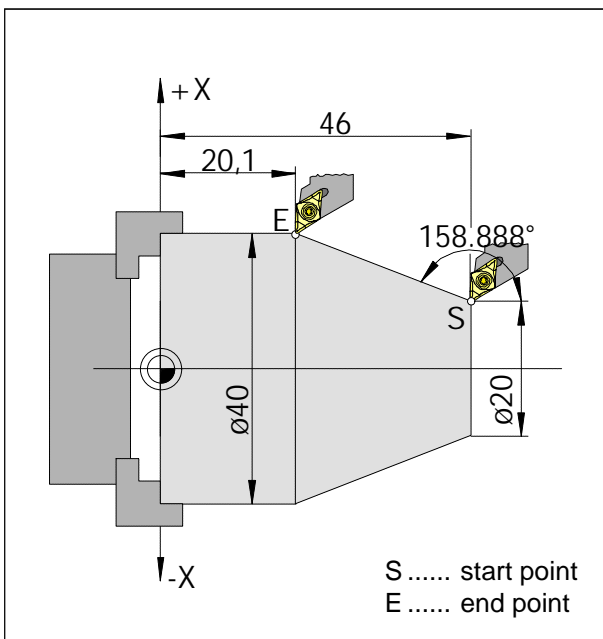
S ...... start point
E ...... end point

Absolute and incremental measures

emco

G02 Circular Interpolation Clockwise

G03 Circular Interpolation Counterclockwise

Format

N... G02/G03 X... Z... I... K... F...
or
N... G02/G03 X... Z... B... F...

X, Z .... End point of the arc (absolute or incremental)

I, K .... Incremental circle parameter (Distance from the start point to centre of arc, I is related to X, K to Z-axis)

B .... Radius of the arc (arc smaller than semicircle with +B, larger than semicircle with -B), can be entered instead of I, K

The tool will be traversed to the end point along the defined arc with the programmed feed F.
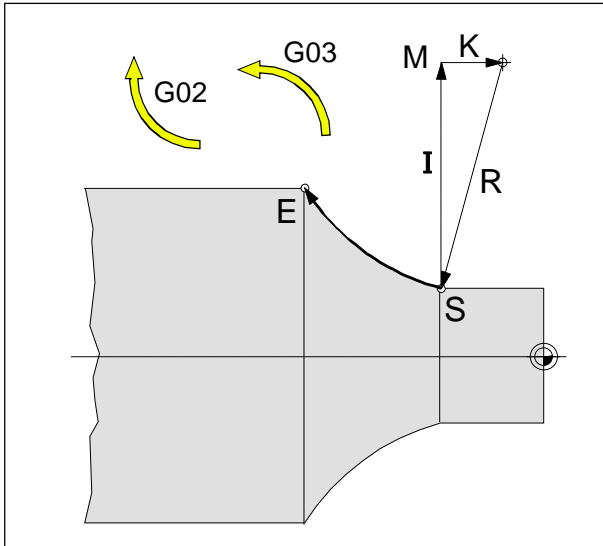
Parameters and rotational direction

Note

Programming the value 0 for I or K can be omitted.
The position of the circle end point will be checked, a tolerance of 100 µm (computing and rounding errors) is allowed.
According to DIN 66025 the observation of G02, G03 is always behind the turning axis, no matter whether machining occurs behind or in front of the turning axis.

emco

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 = 2.5sec)

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.

| Exact stop active | Exact stop not active |
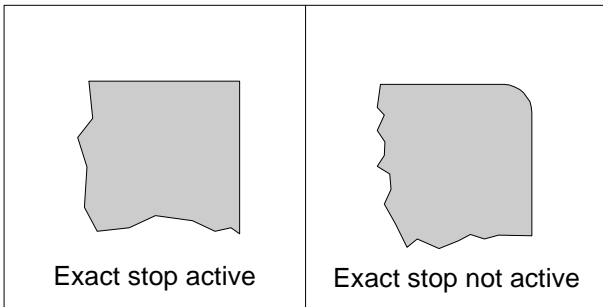
G10 - G13 Polar Coordinate Interpolation

G10 Positioning (rapid traverse)
G11 Linear interpolation
G12 Circular interpolation clockwise
G13 Circular interpolation counterclockwise

With angle and radius dimensioned drawings can be entered directly with polar coordinates.
To determine the traverse path the control needs the centre point, the radius and the angle.
The centre point will be entered 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 B.
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).

G10 X1 Z1 A1 B
G11 A2
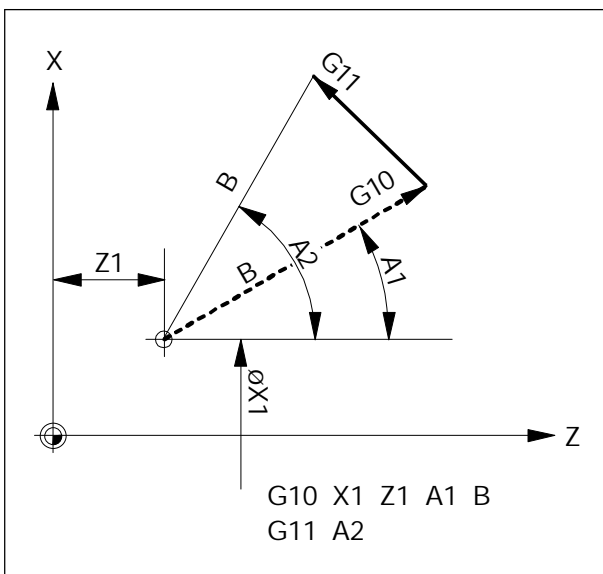
Movements determined by polar coordinates

EMCO
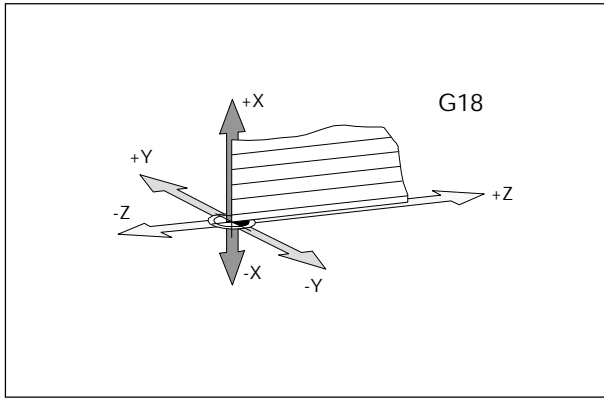
G16, G18 Working plane selection

Format
N... G16 ZXZX
N... G18

G16   Plane selection with free axis selection
G18   Plane ZX

G16 / G18 selects the working plane.

• The tool axis is vertical to the working plane.
• Circle interpolation G2/G3 occurs in the working
  plane.
• Polar coordinate interpolation occurs in the working
  plane.
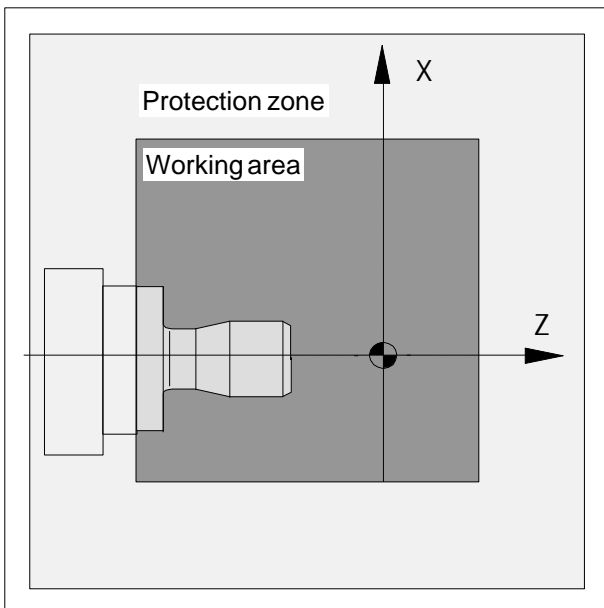
G25/G26 Programmable working area limitation

Format::
N... G25 X... Z...
N... G26 X... Z...

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.

emco
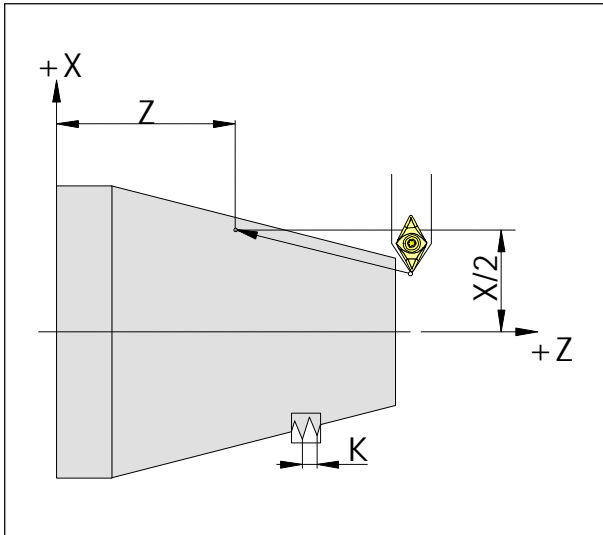
Measures for thread cutting

G33 Thread Cutting

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

I/K ....Thread pitch [mm]
I ....in X direction (longitudinal)
K ....in Z direction (face)

Straight, tapered and scroll threads can be cut.
For thread pitch I or K has to be entered according to the main direction of the thread (longitudinal or face).
Machining routines like knurling are also possible.

Note
- Feed and spindle override are not active with G33 (100%).
- A fair sized undercut has to be machined first.
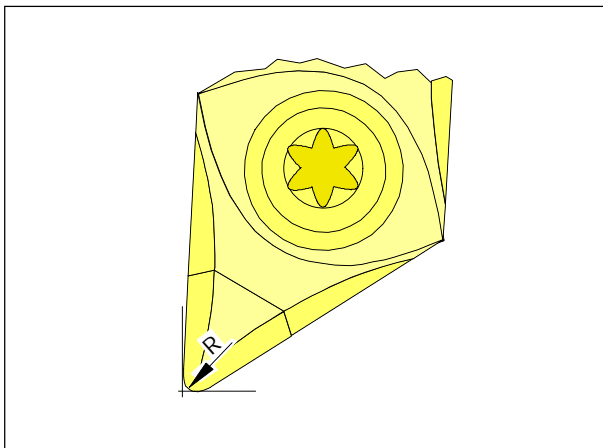
Tip radius and theoretical cutter tip

Cutter Radius Compensation

During tool measurement the tool tip is merely measured at two points (touching the X and Z axes).

The tool offset therefore only describes a theoretical cutter tip.

This point is traversed on the workpiece in the programmed path.

With movements in the axis directions (longitudinal and face turning) the points on the tool tip touching the axes are used.
No dimensional errors are produced on the workpiece.

With simultaneous movements in both axis directions (tapers, radii) the position of the theoretical cutting point no longer coincides with the point on the tip actually cutting. Dimensional errors occur on the workpiece.
Maximum contour error without cutter radius compensation with 45° movements:

Cutter radius 0,4 mm ≙ 0,16 mm path distance

≙ 0,24 mm distance in X and Z.

If cutter radius compensation is used, these dimensional errors are automatically calculated and compensated by the control.

Cutting movements parallel to the axes and slant

emco

Tool pathes with selection / cancellation of the cutter radius compensation
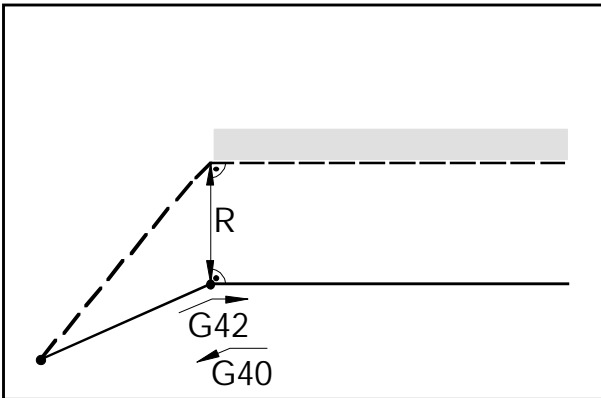
Frontal approach or leaving of an edge point
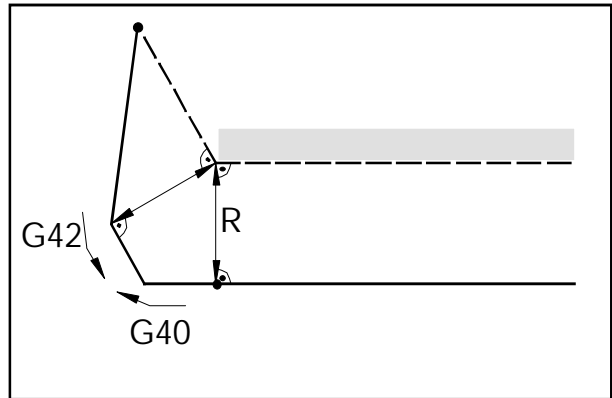
Approach or leaving of an edge point side behind
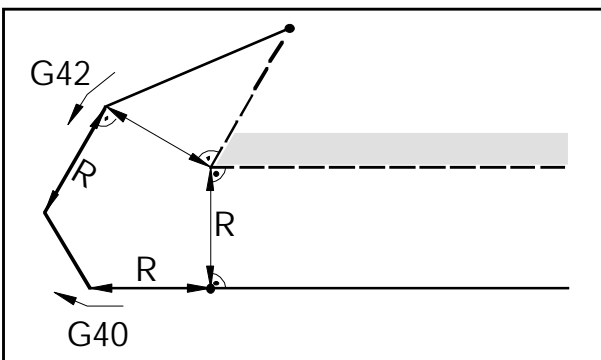
Approach or leaving of an edge point behind

— — — 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 tip 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
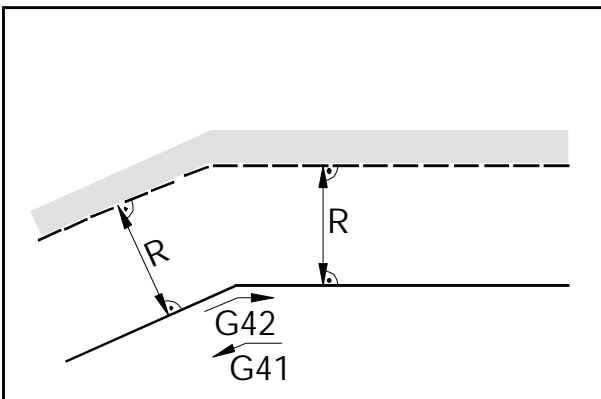
Tool path at an internal edge
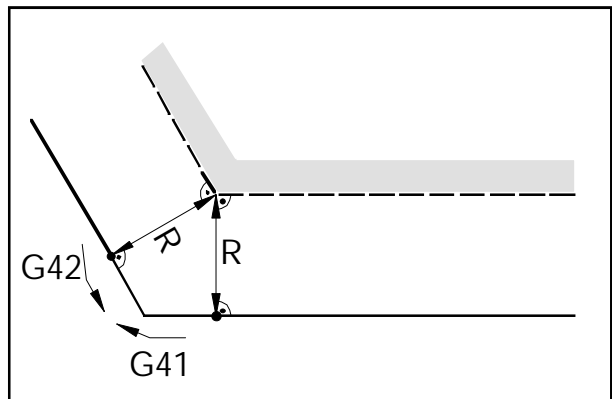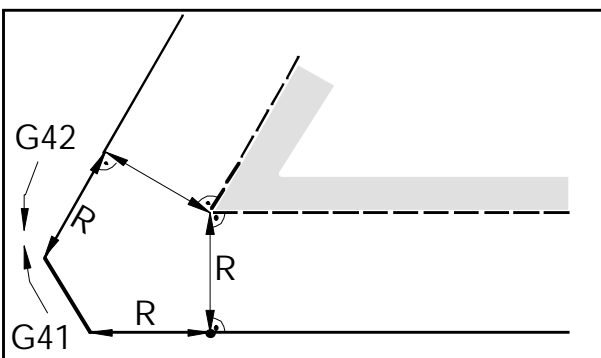
Tool path at an outer edge > 90°

— — — 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 tip 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 path at an outer edge < 90°

G40 Cancel Cutter Radius Compensation

The cutter radius compensation will be cancelled by G40.
Cancellation is only permitted with a linear traversing command (G00, G01).
G00 or G01 can be programmed in the same block or as the first traversing movement after cancellation.
G40 is programmed frequently in the block with return to the tool change position.
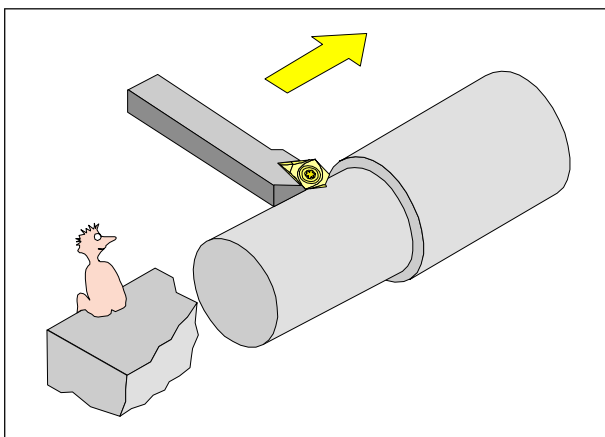
Definition G41 cutter radius compensation left

G41 Cutter Radius Compensation Left

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

Note
- No direct change between G41 and G42 - cancel with G40 previously.
- Cutter radius R and cutter position (tool type) must be defined.
- Selection is only permitted in conjunction with G00 or G01.
- Change of the tool correction is not possible while cutter radius compensation.
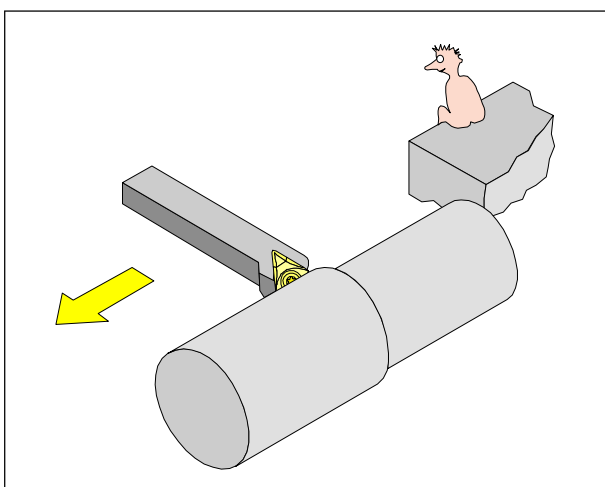
Definition G42 cutter radius compensation right

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 cutter radius is to be selected with G42.

Note see G41!

ƎMCO

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
G247 Soft approach to contour with quarter circle
G347 Soft approach to contour with semicircle
G48   Leave as approached
G148 Soft leaving the contour with linear
G248 Soft leaving the contour with quarter circle
G348 Soft leaving the contour 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.
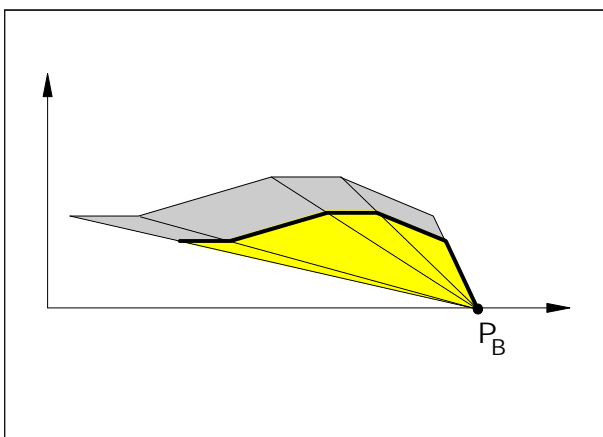
G50   Cancel Scale Modification

G51   Scale Modification

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

With X and Z the base point (P_B) for scale modification is determined, with P the scale factor.
If X and Z are not determined, the work piece zero point (X = 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

emco

G53 Cancel Zero Offset Blockwise

Format:
N... G53

The machine zero point is determined by the machine manufacturer (EMCO lathes: on the rotating axis and on the end face of the spindle flange).
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 point.

G54 - G57 Zero Offset 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 - G57.
G54 is initial status and active without call-up.

G58/G59 Programmable Zero Offsets

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

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

Normally with G58/G59 the workpiece length will be considered.
With A a coordinate rotation can be programmed.

With program end, program abortion and RESET G58/G59 will be deleted.
The commands G58 and G59 are effective blockwise, the zero offset caused by G58/G59 is active, until it will be altered or cancelled.

EMCO

| Exact stop active | Exact stop not active |

Speed characteristics of the slides with G62/G64

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 transititions will be exact.
G60 is active until it will be cancelled by G62 or G64.

G62, G64 Deselection Exact Stop Mode

Format
N... G62/64

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

G63 Thread tapping with compenstion chuck

Format

G63   X..   Z..   F..   S..

Thread tapping without synchronisation.

The spindle speed is programmed under the address S and a suitable feedrate underthe address F.

G63 is effective blockwise. While G63 the feed and spindle override are locked to 100%.
The retraction movement (with opposite spindle direction) also must be programmed with G63.

Example:
Tap M5 (pitch P = 0,8 mm)
spindle speed S = 200, therefore F = 160

N10   G1   X0   Z3   S200   F1000   M3
      (approach start point)
N20   G63   Z-50   F160
      (therad tapping, thread depth 50)
N30   G63   Z3   M4
      (retraction, change of spindle direction)

EMCO

G70   Measuring in Inches

Format
N5 G70

By programming the following data will be transformed into the inch system:
• Path information X, Z
• Interpolation parameter I, K
• Chamfers, radii +B, -B

Notes
• For clearness G70 should be defined in the first block of the program.
• A change between G70 and G71 within a program is 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 Millimetre

Format
N5 G71

Comment and notes like G70!

EMCO

G90   Absolute Programming

Format
N... G90

The addresses have to be programmed as following:
X .... diameter
Z+/- ...... absolute (referred to the workpiece zero point)

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

G91 Incremental Programming

Format
N... G91

The addresses have to be programmed as following:
X .... radius
Z .... Incremental (real) traverse path with sign

Note see G90.

G92   Spindle Speed Limit

Format
N... G92 S...   [U/min]

With G92 a maximum spindle speed (rev/min) can be determined for a part program (effective only in connection with G96).
Enter the desired maximum speed with address S.
This command is used in conjunction with constant cutting speed.

EMCO
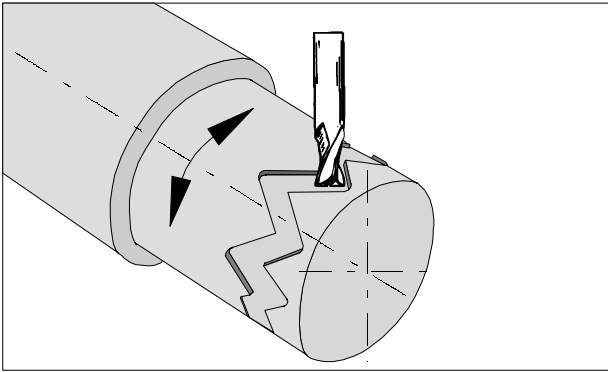
G 92 Cylindrical interpolation

Format
N... G92 P.. C

G92 P.. ......Cylindrical interpolation ON
G92 P1 ...... Cylindrical interpolation OFF
P .... Factor for unit circle
C................ 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

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

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

emco
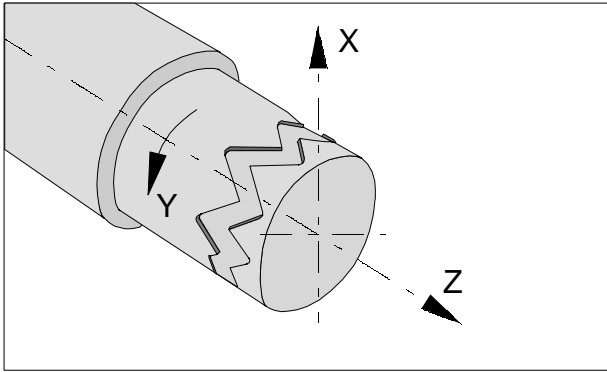
Example Cylindrical interpolation

```
G54
G58 Z40
T7 D7
F200 S2=2000 G94 M2=3
M52          Couple in and Positioning of the spindle
G92 P0.33 C    Cylindrical interpolation ON.
               Unit diameter = 360/PI = 114.59
                P =38.2 /114.59= 0.33
                 C = Axis name for round axis
G0 X45 Z-5
G1 X35 C0 Z-5
G1 Z-15 C22.5
Z-5 C45
Z-15 C67.5
Z-5 C90
Z-15 C112.5
Z-5 C135
Z-15 C157.5
Z-5 C180
Z-15 C202.5
Z-5 C225
Z-15 C247.5
Z-5 C270
Z-15 C292.5
Z-5 C315
Z-15 C337.5
Z-5 C360
X45
G92 P1            Cylindrical interpolation OFF
M53               C- axis OFF
G0 X80 Z100
M2=5              Spindle for driven tool  OFF
M30
```

Milled with end milling cutter ⌀5mm
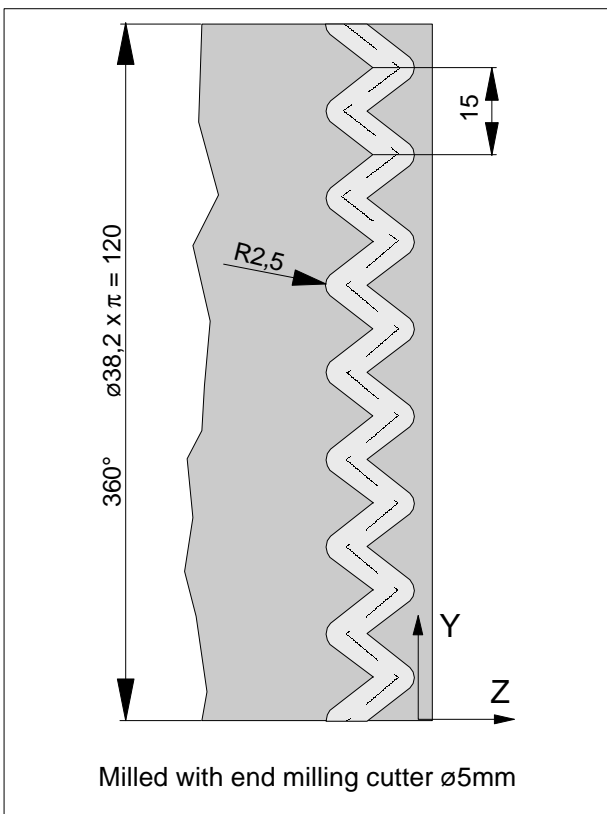
G94   Feed per Minute

By G94 all F (feed) values are in mm/min (inch/min).

G95   Feed per Revolution

By G95 all F (feed) values are in mm/revolution (inch/revolution).

G96   Constant Cutting Speed

Unit: m/min feet/min
The control computes the spindle speed for the actual diameter continuously.
With diameters decreasing to 0 the speed should increase to infinite.
In fact it increases to the maximum speed of the machine and the program runs on without alarm.
If the maximum speed is to high (e.g.: limited chuck speed, unbalanced work pieces, ...) G92 has to be programmed additionally to delimit the speed.
Feed is automatically set to G95 (mm/rev, inch/rev).
No zero offset in X direction must be active.

G97   Constant Spindle Speed

Unit: rev/min
G96 will be deselected and the spindle speed will be holded at the last valid value. Afterwards S will be programmed in rev/min.

EMCO
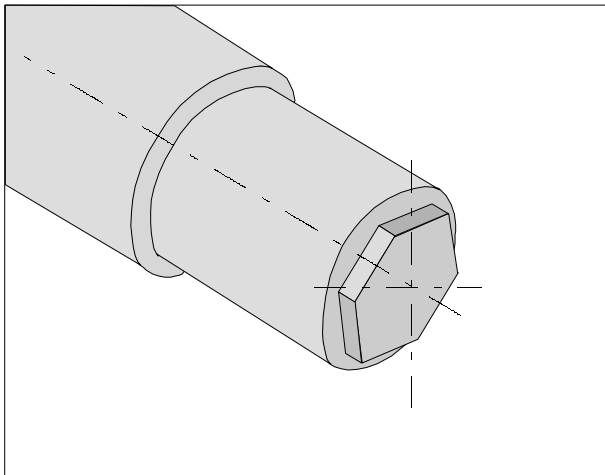
G131 Transmit

TRANSMIT - TRANSform - **M**illing **I**nto **T**urning

The polar coordinate interpolation is adequate for machining the end face of a turned part.

Format:

G130 .... Transmit OFF
G131 .... Transmit ON
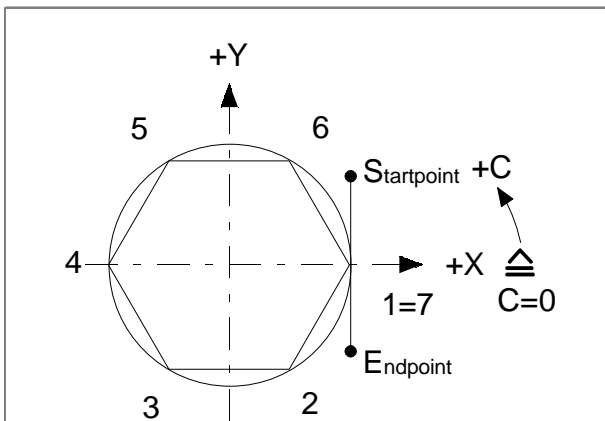
Point	X	Y
S	17.32	10
1	17.32	0
2	8.66	-15
3	-8.66	-15
4	-17.32	0
5	-8.66	15
6	8.66	15
7	17.32	0
E	17.32	-10

Example- Transmit

G54	
G58 Z13	
T3 D3	
M52	Couple in and positioning of the spindle
G94 F200 S2=2000 M2=3	Spindle for driven tools on
G16 Z X Z X	Identify axis for lengh correction. (Attention, L1=Z ; L2=X)
G131	Transmit ON
G0 X1=30 Y1=10 Z-6	
G1 X1=17.32 Y1=10 G41	
Y1=0	
X1=8.66 Y1=-15	
X1=-8.66 Y1=-15	
X1=-17.32 Y1=0	
X1=-8.66 Y1=15	
X1=8.66 Y1=15	
X1=17.32 Y1=0	
Y1=-10	
X1=25 Y1=-10 G40	
G130	End of Transmit
M53	C- axis OFF
M2=5	Spindle for driven tool OFF
M30	
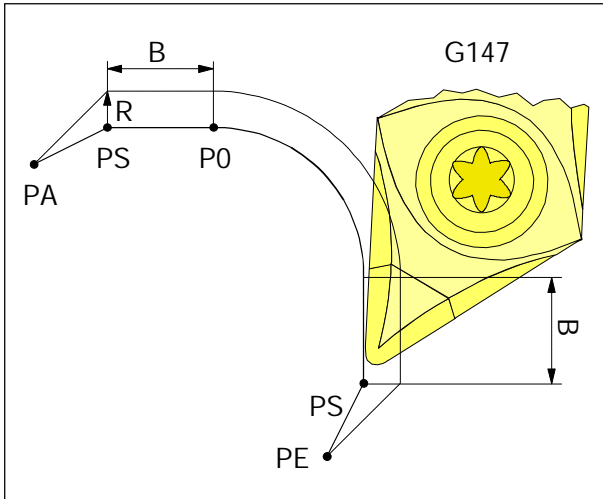
Soft approach and leaving with linear
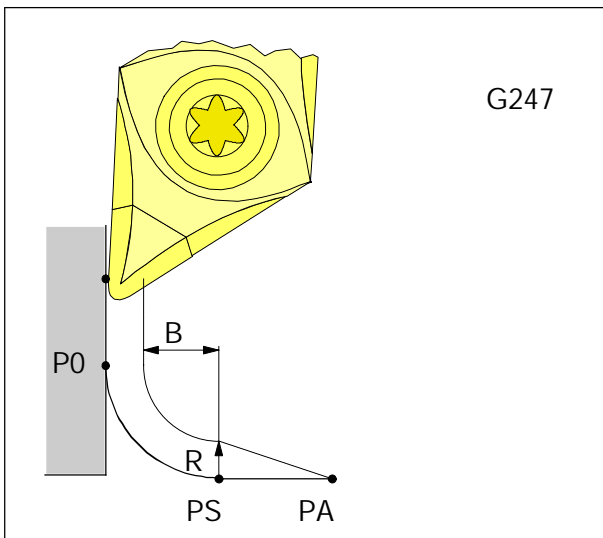
Soft approach and leaving with quarter circle
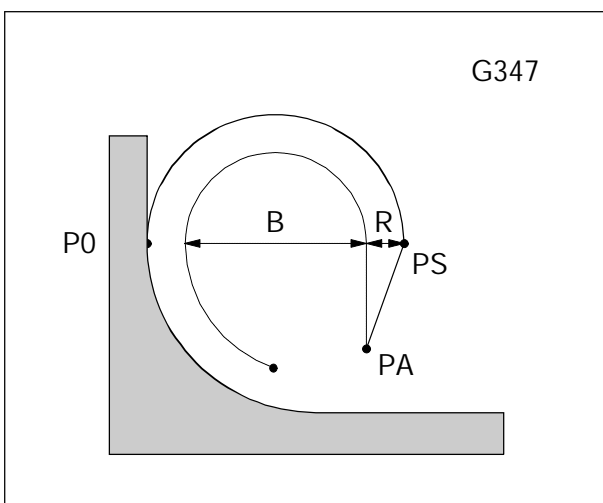
Soft approach and leaving with semicircle

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 with Linear
G248 Soft Leaving with Quarter Circle
G348 Soft Leaving with Semicircle

Format
N5 G147/247/347/148/248/348 X... Z... B...

- This 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
  the value B (approaching distance without contour touch)

- In the leaving block must be determined:
  the coordinates of the start point PE after leaving the contour
  the value B (leaving distance without contour touch)

- In an approaching or leaving block no further traverse movements must be programmed.

- After an approaching or leaving block no block must be programmed which contains only auxiliary functions.

- G41 or G42 must be active before an approaching block.

- In the leaving block G40 will be set automatically, that means, G41 or G42 have to be programmed again.

- Soft approaching or leaving is not possible with contours which were programmed with contour drafts.

PA Start point before approaching the contour
PS Supporting point, will be calculated by the control basing on B
P0 End point of the approaching block = start point of the contour
PE End point after leaving the contour
R   Tool radius
B   Distance without contour touch
dashed line: path of the tool tip radius centre

Description of M Commands

M Commands are switching or addidional functions (miscalleneous). The M commands can stand alone in a program block or together with other commands. Commands of the same group cancel each other, that means the M command programmed last cancels the previously programmed M command of the same group.

Remark
The following pages describe the standard M commands. Wheter these M commands are executable depends on the type of the machine and the used accessories.

M00 Programmed Stop Unconditional

These command effects a stop in the execution of the part program.
Main spindle, feed and coolant will be switched off. The chip protection door can be opened without triggering 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, but only if the function PRO-GRAMMED STOP YES was switched on by softkey in the menu PROGRAM CONTROL.

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 Main Spindle ON Clockwise

The spindle is switched on provided that a spindle speed or cutting speed has been programmed, the chip protection door is closed and a workpiece correctly clamped.
M03 must be used for all right-hand cutting or over-head clamped tools, if machining occurs behind the turning centre.

M04 Main Spindle ON Counterclockwise

The same conditions as described under M03 apply here.
M03 must be used for all left-hand cutting or normal clamped tools, if machining occurs behind the turning centre.

M05 Main Spindle Off

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

EMCO

M08 Coolant ON

only for EMCO PC Turn 120.
The coolant will be switched on.

M09 Coolant OFF

only for EMCO PC Turn 120/125.
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).

M20 Tailstock BACK

only for accessory automatical tailstock.
The tailstock moves back.
See G: Accessory Functions - Automatic Tailstock

M21 Tailstock FORWARD

only for accessory automatical tailstock.
The tailstock moves forward.
See G: Accessory Functions - Automatic Tailstock

M25 Open Clamping Device

only for accessory automatical clamping device on the PC TURN 120.
The clamping device opens.
See G: Accessory Functions - Automatic Clamping Device

M26 Close Clamping Device

only for accessory automatical clamping device on the PC TURN 120.
The clamping device closes.
See G: Accessory Functions - Automatic Clamping Device

M30 Main Program End

With M30 all drives are switched off and the control is returned to the start of the program.
Moreover, the counter level is increased by 1.

M52 C- Axis ON

only for machines with driven tools.
Machine are switched in the round axis operation.

M53 C- Axis OFF

only for machines with driven tools.
The round axis operation of the machine are switched off.

M71 Puff Blowing ON

only for accessory puff blowing device
The puff blowing device will be switched on.

M72 Puff Blowing OFF

only for accessory puff blowing device
The puff blowing device will be switched off.

EMCO

Description of Cycles

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

Example
N... R20=... R21=... R22=... R24=... R25=...
    R26=... R27=... R28=... R29=... R30=...
    L95   P2
That means that the cycle L95 with the programmed parameters will run 2 times.

L93   Cut-in Cycle

The cut-in cycle allows machining of grooves longitudinal, face, outside and inside.

Note:
Both cutting edges of the cut-in tool have to be measured in **neighbouring** tool data registers (e.g.: D21 and D22).

Programming the parameter:

R10   0 for longitudinal, 1 for face cut-in
R21   Outer resp. inner diameter
R22   Start point in Z
R23   Determine the start point
      Longitudinal cut-in
      1   inside/outside right
      -1   inside/outside left
      Face cut-in
      1   inside right/left
      -1   outside right left
R24   Finishing offset in X
R25   Finishing offset in Z
R26   Infeed depth
R27   Width of cut-in
R28   Dwell of the bottom of the cut-in
R29   Angle left flank (0° - 89°)
R30   Radius (+) or chamfer (-) at the bottom left
R31   Cut-in diameter
R32   Radius or chamfer at edge of cut-in left
R33   Radius or chamfer at bottom of cut-in right
R34   Radius or chamfer at edge of cut-in right
R35   Angle right flank
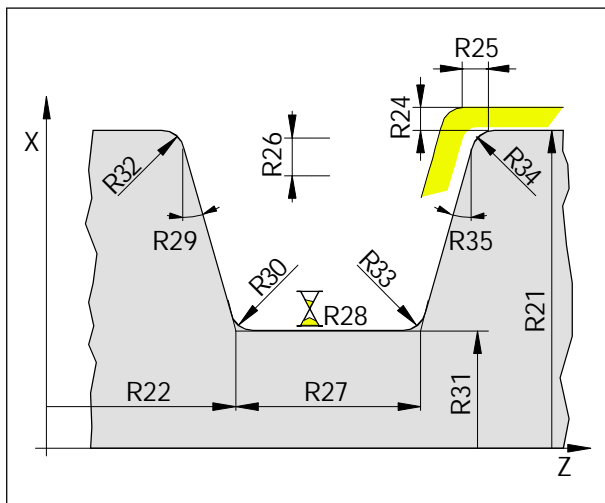
Dimensions for cut-in

L94   Undercut Cycle

The undercut cycle L94 allows undercuts with standard stress according to DIN 509, form E and F with a finished part diameter > 18 mm.
The cutter radius compensation will automatically be activated in the cycle.
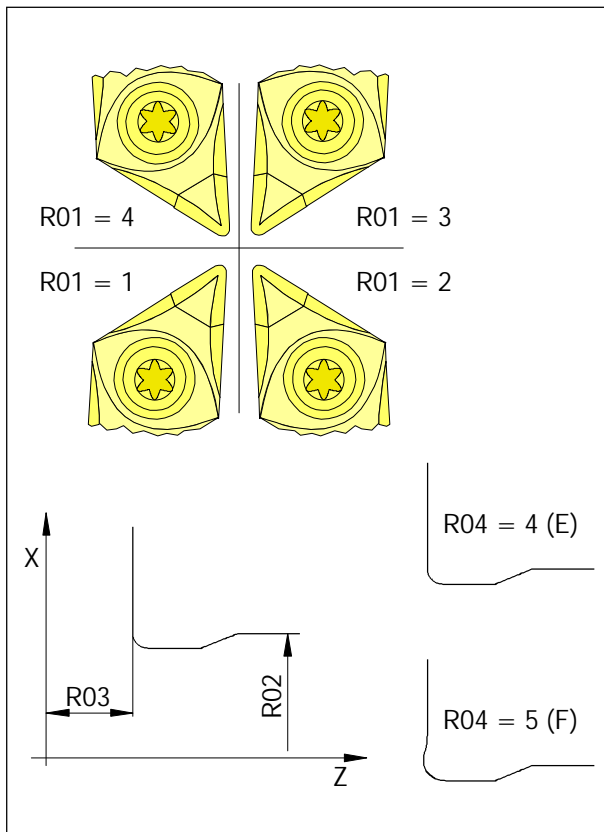
Dimensions for undercut

Programming the parameter:

R01   Definition of the cutter position (1 - 4)
R02   Start point of the contour in X
      R02 defines the diameter of the finished part. The cycle adds automatically 2 mm to this value, this position is the start point in X.
R03   Start point of the contour in Z
      For R03 the finished part dimension will be entered. The cycle adds automatically 10 mm to this value, this position is the start point in Z.
R04   Sign for form E or F
      R04=4 Form E for workpieces with one machining side
      R04=5 Form F for workpieces with two machining sides which are perpendicular to each other.

emco

L95   Stock Removal with Back Pockets

L96   Stock Removal without Back Pockets

Machining can be programmed longitudinal, on face, inside, outside.

For L95 max. 10 back pockets (in machining direction decreasing diameters) may be programmed.

The **first** contour point will be determined in the circle. The contour draft has to be described in a subroutine (G1, G2, G3).

Dimensions for the stock removal cycle

The **last** point in the subroutine determines the start diameter of the roughing sequence. For face operation this point must be the highest point in X, for longitudinal operation this must be the lowest point in Z.

The **first** block in the subroutine must be programmed in absolute coordinates.

Programming the parameter:

R20   Selection of the desired contour (subroutine number)
R21   Start point of the contour in X
R22   Start point of the contour in Z
R24   Finishing offset in X
R25   Finishing offset in Z
      R24 and R25 are valid for roughing and finishing, with R24=0 and R25=0 the finish dimension will be reached
R26   Roughing infeed (X or Z), will be omitted with finishing
R27   Selection cutter radius compensation (40, 41, 42), the control activates the cutter radius compensation automatically, correct selection and deselection automatically
R28   Feed
R29   Machining procedure, see table
R30   Feed factor with back pockets
      With this number (e.g.: 0,7) the feed will be multiplied for dive in movements with back pockets (feed reduction)

R29=XX	1X	2X	3X	4X
X1	Roughing axis parallel longitudinal outside	Finishing to finishing offset longitudinal outside	Roughing axis parallel and one roughing cut contour parallel (rest edges) longitudinal outside	Complete machining (roughing, rest eedges, finishing) longitudinal outside
X2	Roughing axis parallel face outside	Finishing to finishing offset face outside	Roughing axis parallel and one roughing cut contour parallel (rest edges) face outside	Complete machining (roughing, rest eedges, finishing) face outside
X3	Roughing axis parallel longitudinal inside	Finishing to finishing offset longitudinal inside	Roughing axis parallel and one roughing cut contour parallel (rest edges) longitudinal inside	Complete machining (roughing, rest eedges, finishing) longitudinal inside
X4	Roughing axis parallel face inside	Finishing to finishing offset face inside	Roughing axis parallel and one roughing cut contour parallel (rest edges) face inside	Complete machining (roughing, rest eedges, finishing) face inside

Machining possibilities with R29

L97 Thread Cutting Cycle

Longitudinal, face, inside and outside threads can be programmed.
The infeed of the tool occurs automatically and is degressively quadratic. This keeps the cross section of the chip constant.
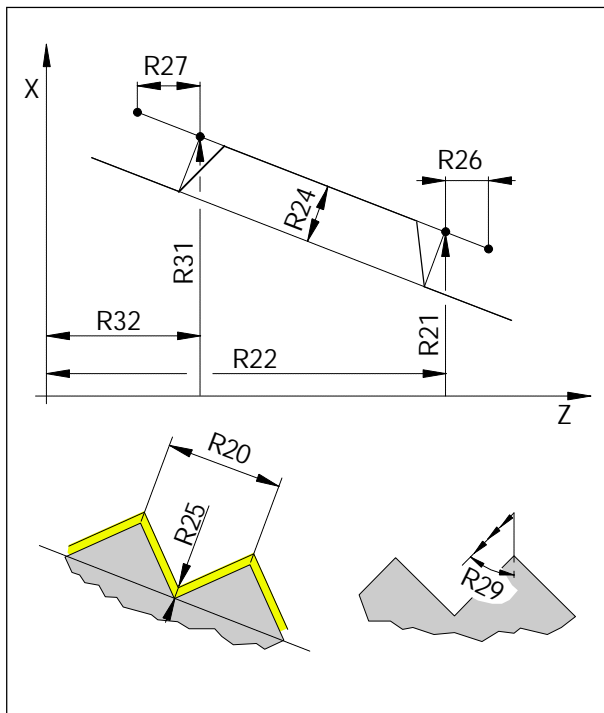
Thread cutting cycle

Programming the parameter:

R20   Thread pitch (always the value parallel to axis)
R21   Start point of the thread in X
R22   Start point of the thread in Z
      R21 and R22 describe the real start point at the contour.
R23   Number of idle cuts
R24   Thread depth (positive value = inside thread, negative value = outside thread)
R25   Finishing offset
      After automatic cut dividing with roughing occurs a finishing cut with the programmed cutting depth.
R26   Thread let-in, incremental without sign
R27   Thread let-out, incremental without sign
      The values R26 and R27 always will be entered parallel to axis without sign. With taper threads the correct starting and end points will be calculated automatically.
R28   Number of roughing cuts
R29   Infeed angle (half flank angle)
      Flank infeed is only possible with longitudinal or face threads (no taper threads)
R31   End point of the thread in X (absolute)
R32   End point of the thread in Z (absolute)
      R31 and R32 describe the real end point at the contour.

L971 Longitudinal Thread

This cycle was created for longitudinal and less tapered threads and is time optimized.
Programming occurs like with L97.

Differences to L97:

R28
pos. sign: degressive infeed
neg. sign: constant infeed

R34
This parameter was inserted additionally. With this parameter a retraction height of the tool over the thread can be programmed.
Inside threads: 0,01 - 1 mm
Outside threads: 0,01 - 3 mm

L98   Deep Hole Drilling Cycle

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

Chip breaking (R11=0)
The drill dives into the work piece to the first drilling depth (R25), dwells (duration R27), retracts 1 mm and dives in again.

Retracting (R11=1)
The drill dives into the work piece to the first drilling depth (R25), dwells (duration R27), retracts out of the drilling hole (to R22), dwells (duration R28), and dives in again.

The following infeed is always smaller for the value R24 than the previous. The sequence infeed - retraction will be repeated until the end depth is reached..
If the calculated infeed is smaller than R24, it will be constant with the value R24.
If the remaining infeed down to end depth R26 is smaller than the double degression value (2xR24), the remaining infeed will be halved and worked off in two infeeds.. Therefore the smallest infeed never can be less than R24/2.
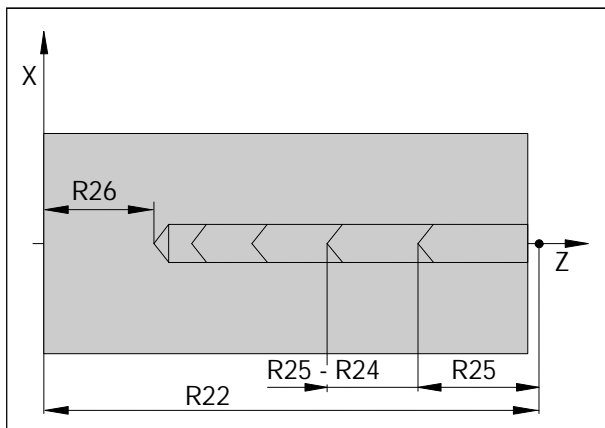
Programming the parameter:

R11   0   chip breaking
1   retracting
R22   Start point in Z (absolute)
R24   Degression value (incremental, without sign)
R25   First drilling depth (incremental, without sign)
R26   End drilling depth (absolute)
R27   Dwell at start point (for retracting only)
R28   Dwell at drilling depth (chip breaking and retracting)

Deep hole drilling cycle

L99   Thread Draft

This cycle is for cutting of following threads (longitudinal or face). The single pitches can be different.

Programming the parameter:

R11   Start point of the thread in X absolute
R12   1. intermediate point in X absolute
R13   2. intermediate point in X absolute
R14   End point of the thread in X absolute
R21   Start point of the thread in Z absolute
R22   1. intermediate point in Z absolute
R23   2. intermediate point in Z absolute
R24   End point of the thread in Z absolute
R25   Finishing offset
      After automatic cut dividing with roughing occurs a finishing cut with the programmed cutting depth.
R26   Tread let-in, incremental without sign
R27   Tread let-out, incremental without sign
      The values R26 and R27 always will be entered parallel to axis without sign. With taper threads the correct starting and end points will be calculated automatically.
R28   Number of roughing cuts
R29   Infeed angle (half flank angle)
      Flank infeed is only possible with longitudinal or face threads (no taper threads)
R35   Number of idle cuts
R36   Thread depth (positive value = inside thread, negative value = outside thread, face thread)
R41   Thread pitch 1
R42   Thread pitch 2
R43   Thread pitch 3

Thread draft

emco
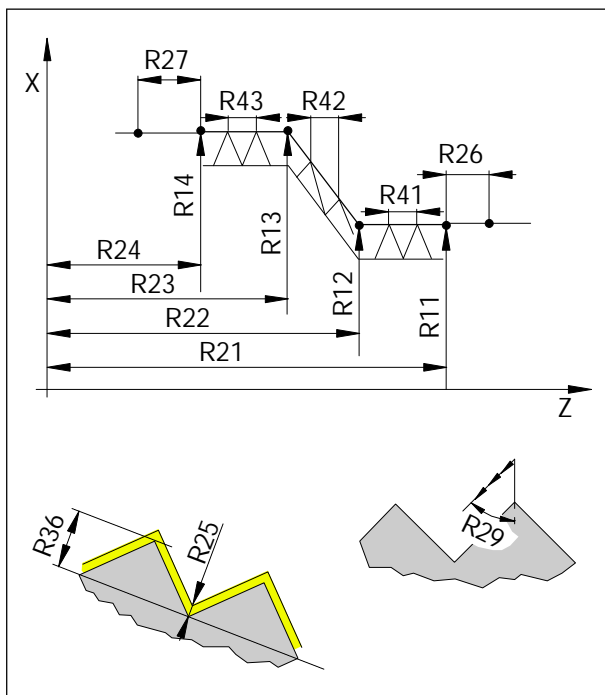
Contour definition

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.
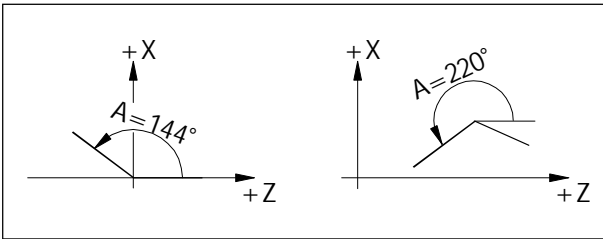
Angles refer to the +Z direction

Angles are always related to the +Z direction.

The following contour drafts are offered:
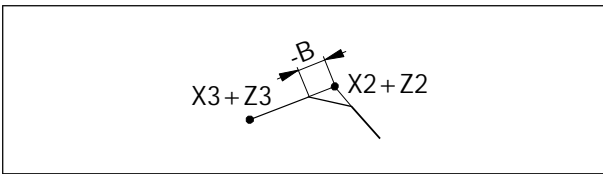
Inserting a chamfer

Insert Chamfer

For chamfers B is entered as negative number.

G1 X2... Z2... B-... LF
G1 X3... Z3... LF
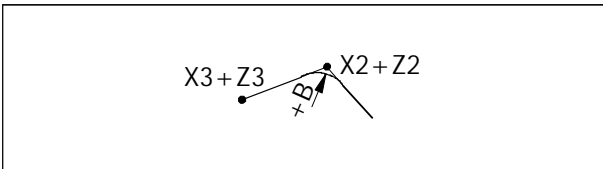
Inserting a radius

Insert Radius

For radii B is entered as positive number. The entered radius must not be larger than the smaller one of the two lines.

G1 X2... Z2... B+... LF
G1 X3... Z3... LF
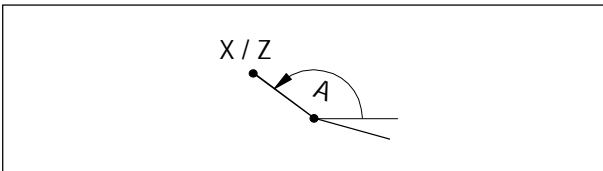
Line determined by angle and one end coordinate

Line

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

G1 A... X... or G1 A... Z...
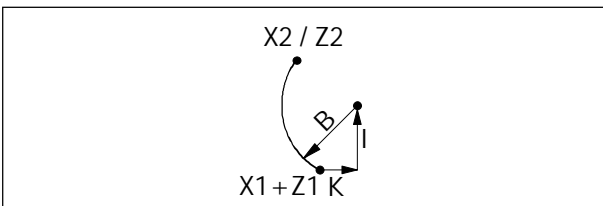
Arc defined by radius, centre and one end coordinate

Arc

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

G2 I... K... B... X2... or G2 I... K... B... Z2...

EMCO

Contour draft line - line

Line - Line

Angle A, one coordinate of the first point and both coordinates of the target point.
Radii +B or chamfers -B can be inserted; if the next following block is a G1 block, they also can be added on.
G1 A... X1...
G1 X2... Z2...
or
G1 A... Z1...
G1 X2... Z2...

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

G1 A1... A2... X2... Z2...
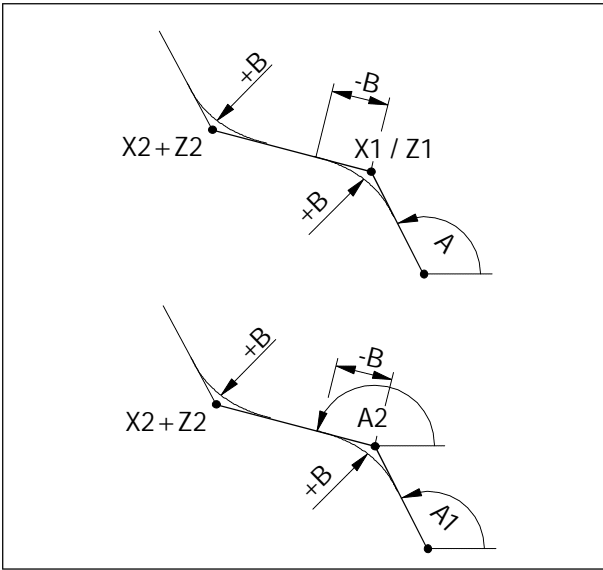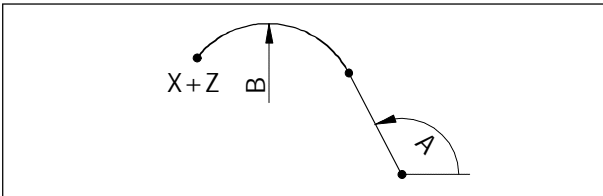
Contour draft line - arc (tangential)

Line - Arc (tangential)

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

G3 A... B... X... Z...
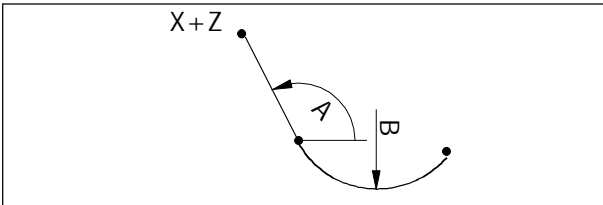
Contour draft arc - line (tangential)

Arc - Line (tangential)

Radius B, angle A and both coordinates of the target points.

G2   B... A... X... Z...
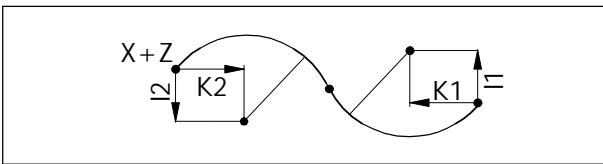
Contour draft arc - arc (tangential)

Arc - Arc (tangential)

The path command G2, G3 will be programmed for the first arc. The second path command is always opposite and will not be programmed. The interpolation parameter 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... Z...

EMCO

Subroutines

Function sequences which are repeated multiple can be programmed as subroutines.
Contour descriptions for cycles also will be entered as subroutine.

The numbers L90 - L100 are reserved for cycles and must not be used for subroutines.
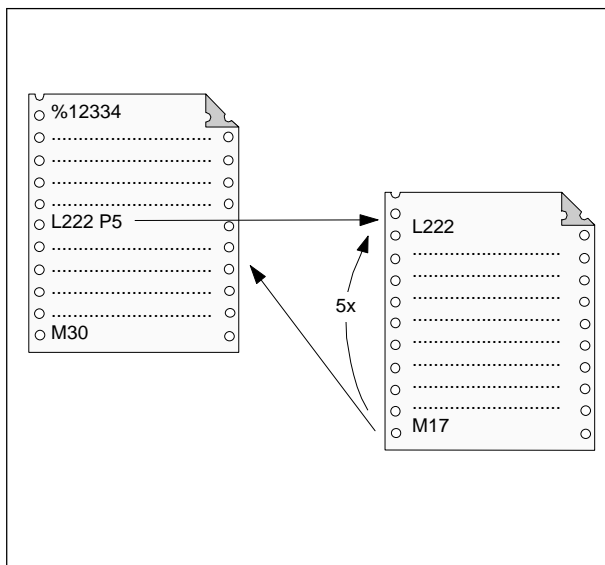
Multiple call of a subroutine

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
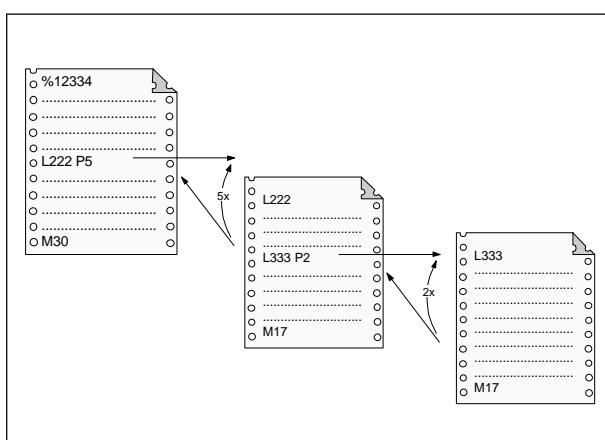
Subroutine Nesting

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

Nesting subroutines

emco

D 32

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

EMCO

@- Befehl		Funktion
@040	<Const><R Par 1> ... <R Par n>	Sichern der angegebenen lokalen R-Parameter auf den Stack
@041	<R Par 1><R Par 2>	Sichern einer Gruppe lokaler R-Parameter auf den Stack
@042	<Const><R Par n> ... <R Par 1>	Gesicherte R-Parameter vom Stack holen
@043	<R Par 1><R Par 2>	Gruppe der gesicherten R-Parameter vom Stack holen
@100 @100	<Const> <R Par>**	Absoluter Sprung zu NC-Satz

@111	<Var> <Wert 1><Const 1>
	     <Wert 2><Const 2>
	     ...
	     ...
	     <Wert n><Const n>
		Case-Verzweigung

@12y	<Var><Wert><Const>	IF-THEN-ELSE-Anweisung y Ñ Vergleichsoperator <Vop> Var Ñ R-Parameter oder Pointer
@13y	<Var><Wert><Const>	Wiederholungs-Anweisung mit Abfrage der Wiederholbedingung am Anfang y Ñ Vergleichsoperator <Vop>
@14y	<Var><Wert><Const>	Wiederholungs-Anweisung mit Abfrage der Wiederholbedingung am Ende y Ñ Vergleichsoperator <Vop>
@151	<Var><Wert 2><Const>	Wiederholungs-Anweisung mit Wiederholungen solange bis <Var> inkrementell <Wert 2> erreicht hat
@161	<Var><Wert 2><Const>	Wiederholungs-Anweisung mit Wiederholungen solange bis <Var> dekrementell <Wert 2> erreicht hat
@200	<Var>	Lösche Variable
@201	<Var><Wert>	Lade Variable mit Wert
@202	<Var 1><Var 2>	Tauschen der Variableninhalte
@210	<Wert 3><Wert 4>	Lösche Eingabezwischenspeicher Wert 3: EZS-Anfangsadresse Wert 4: EZS-Endadresse
@300	<Var><Wert 1>	Maschinendaten NC Wert 1: Adresse 0 ... 4999
@302	<Var><Wert 1><Wert 2>	Maschinendaten NC-Bits Wert 1: Byte-Adresse 5000 ... 6999 Wert 2: Bit-Adresse 0 ... 7
@310	<Var><Wert 1>	Settingdaten NC Wert 1: Adresse 0 ... 4999
@320	<Var><Wert 1><Wert 2><Wert 3>	Werkzeugkorrektur Wert 1: 0 Wert 2: D-Nr. 1 ... 99 Wert 3: P-Nr. 0 ... (9)
@330	<Var><Wert 1><Wert 2><Wert 3>	Einstellbare Nullpunktverschiebung (G54 - G57) Wert 1: Gruppe 1 ... 4 (G54 - G57) Wert 2: Achs-Nr. 1, 2, ... Wert 3: 0/1 (Grob/Fein)
@331	<Var><Wert 1><Wert 2>	Programmierbare Nullpunktverschiebung (G58, G59) Wert 1: Gruppe 1 oder 2 (G58 oder G59) Wert 2: Achs-Nr. 1, 2, ...
@336	<Var><Wert 2>	Summenverschiebung Wert 2: Achs-Nr. 1, 2,...
@342	<Var><Wert 1><Wert 3>	Programmierte Spindeldrehzahl lesen Wert 1: Kanal-Nr. 0 ... 3 Wert 2: Spindel-Nr. 1 ... 6

@- Befehl		Funktion
@345	\<Var\>\<Wert 1\>\<Wert 2\>	Programmierte Schnittgeschwindigkeit Wert 1: Kanal-Nr. 0, 1, 2 Wert 2: 0 = G96
@360	\<Var\>\<Wert 2\>	Achs-Position-Ist werkstückbezogen Wert 2: Achs-Nr. 1, 2, ...
@361	\<Var\>\<Wert 2\>	Achs-Position-Ist maschinenbezogen Wert 2: Achs-Nr. 1, 2, ...
@363	\<Var\>\<Wert 2\>	Spindelpositions-Istwert Wert 2: Spindel-Nr. 1 ... 6
@364	\<Var\>\<Wert 2\>	Spindeldrehzahl-Istwert Wert 2: Spindel-Nr. 1 ... 6
@367	\<Var\>\<Wert 1\>	Achsnummer der aktuellen Ebene/Leitspindelnummer in R-Parameter \<Var\> lesen: Var+0: Nr. der waagrechten Achse Var+1: Nr. der senkrechten Achse Var+2: Nr. der Achse senkrecht zur Ebene Var+3: Nr. der waagrechten Achse Var+4: Nr. der Achse, in der Länge 2 wirkt (Werkzeugtyp 30) Wert 1: Kanal-Nr. 0, 1, 2
@36a	\<Var\>\<Wert 1\>	D-Funktion Ist Wert 1 = 0
@36b	\<Var\>\<Wert 1\>\<Wert 3\>	Lesen der G-Funktion aus dem Arbeitsspeicher des aktuellen Satzes Wert 1: Kanal-Nr. 0, 1, 2 Wert 3: Interne G-Gruppe, zu der G-Funktion gehört 0 ... 15
@371	\<Var\>\<Wert 1\>\<Wert 3\>	Sonderbits Wert 1: Kanal-Nr. 0 .. 2 = kanalabhängig, 99 = kanalunabhängig
@3e4	\<Var\>\<Wert 1\>	Aktive Getriebestufe lesen Wert 1: Spindel-Nr. 0 bis 6
@420	\<Wert1\>\<Wert2\>\<Wert3\>\<Wert\>	Werkzeugkorrektur Wert 1: 0 Wert 2: D-Nr. 1 ... 99 Wert 3: P-Nr. 0 ... 7 (9)
@423	\<Wert1\>\<Wert2\>\<Wert3\>\<Wert\>	Werkzeugkorrektur additiv Wert 1: 0 Wert 2: D-Nr. 1 ... 99 Wert 3: P-Nr. 0 ... 7 (9)
@430	\<Wert1\>\<Wert2\>\<Wert3\>\<Wert\>	Einstellbare Nullpunktverschiebung additiv Wert 1: Gruppe 1 ... 4 (G54 - G57) Wert 2: Achs-Nr. 1, 2, ... Wert 3: 0/1 (Grob/Fein)
@431	\<Wert1\>\<Wert2\>\<Wert3\>\<Wert\>	Einstellbare Nullpunktverschiebung (G54 - G57) Wert 1: Gruppe 1 ... 4 (G54 - G57) Wert 2: Achs-Nr. 1, 2, ... Wert 3: 0/1 (Grob/Fein)
@432	\<Wert1\>\<Wert2\>\<Wert\>	Programmierbare Nullpunktverschiebung (G58, G59) Wert 1: Gruppe 1 oder 2 (G58 oder G59) Wert 2: Achs-Nr. 1, 2, ...
@440	\<Wert3\>\<Wert\>	Programmierte Achsposition Wert 3: Achs-Nr. 1, 2, ...
@442	\<Wert3\>\<Wert\>	Programmierte Spindeldrehzahl Wert 3: Spindel-NR. 0 ... 6
@446	\<Wert\>	Programmierter Radius
@447	\<Wert\>	Programmierter Winkel

EMCO

@- Befehl		Funktion
@448	<Wert3><Wert>	Programmierter Interpolationsparameter für Kreis und Gewinde Wert 3: Achs-Nr. 1, 2, ...
@4e1	<Wert1><Wert2><Wert>	Spindel-Beschleunigungskonstante schreiben Wert 1: Spindel-Nr. 0 bis 6 Wert 2: Getriebestufe 1 bis 8 Wert 3: Spindelbeschleunigungszeitkonstante 0 bis 16000
	<Var> = <Wert 1> + <Wert 2>	Addition
	<Var> = <Wert 1> - <Wert 2>	Subtraktion
	<Var> = <Wert 1> x <Wert 2>	Multiplikation
	<Var> = <Wert 1> / <Wert 2>	Division
@610	<Var><Wert>	Betragsbildung
@613	<Var><Wert>	Quadratwurzel
@614	<Var><Wert 1><Wert 2>	Wurzel aus Quadratsumme
@620	<Var>	Inkrementiern von <Var> mit 1
@621	<Var>	Dekrementiern von <Var> mit 1
@622	<Var>	Ganzzahliger Anteil
@630	<Var><Wert>	Sinus
@631	<Var><Wert>	Cosinus
@632	<Var><Wert>	Tangens
@634	<Var><Wert>	Arcus Sinus
@637	<Var><Wert 1><Wert 2>	Winkel aus zwei Vektorkomponenten
@640	<Var><Wert>	nat. Logarithmus
@641	<Var><Wert>	e Exponentialfunktion
@710	<Var 1><Var 2>	Referenzaufbereitung Var 1: Ausg.daten ab Var 1 Var 2: Eing.datum ab Var 2
@711	<Var 1><Var 2><Var 31>	Schnittpunktberechnung Var 1: Ausg.daten ab Var 1 Var 2: Erste Kontur ab Var 2 Var 3: mit 0 vorbesetzen
@713	<Var>	Startvorbereitung für Zyklen Var: Ausgangsdaten ab Var
@714		Stop der Decodierung, bis Zwischenspeicher leer ist

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 the 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, machine data, R parameters, tool offset, PLC program, PLC operator messages, PLC machine data, user programs) are read in from an external device (e.g. punched tape reader) or output 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. 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 single. The function is activatet 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 active.

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.

ETX 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".

GENERAL DATA
Select the path for workpiece programs and actual language..

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.

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"

EMCO

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, M word in the buffer memory.

PART PROGRAM
The „PART PROGRAM" softkey leads to:
• EDIT
• CORR BLOCK
• DIRECTORY
• PROG HANDL
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 the PLC bits.

PLC M-DATA
not active

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 byte
• A = Output byte
• M = Flag byte
• T = Timer
• Z = Counter
• DB = Data block
• DW = Data word

PROG. HANDL.
(Program handling)
You can copy, rename and delete the program 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.
(Programmmable zero offset)
Display or entering 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
Input of the coordinate rotation for the single zero offsets G54-G57.

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.

EMCO

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
- 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
In "SETTING DATA": Enter the gear position of the PC TURN 50 with SPINDLE.

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 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").

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

SUBROUTINE
(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.

EMCO

WORKING AREALIM
(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 input as setting data. Programmable and external zero offsets are displayed on the screen.
After "DATA OUT" or "DATA IMPORT" you can select input or output of zero offsets with "ZERO OFFSET".

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 810T

START

SKIP YES-NO | DRY YES-NO | OPT. STP YES-NO | OVERR. YES-NO | DEC.-SBL YES-NO | DRF YES-NO | DRF

STORE X-AXIS | STORE Z-AXIS

NC-ALARM | PLC-ALARM | PLC-MESSAGE | PLC-STATUS | SW VERSION | NC MD | PLC MD | SERVICE AXIS | SERVICE SPINDLE | SERVICE DRIVE

GENERAL DATA | AXIAL DATA 1 | AXIAL DATA 2 | SPINDLE DATA | MACHINE BITS

TOOL OFFSET | SETTING DATA | DATA IN-OUT | PART-PROGRAM | ACTUAL BLOCK | BLOCK SEARCH | PROGRAM CONTROL | DIAG-NOSIS | OVER-STORE | OPS

AUTOMATIC

TOOL OFFSET | SETTING DATA | DATA IN-OUT | PART-PROGRAM | DIAG-NOSIS | TO AUTOM | ZO AUTOM | HAND WHEEL | OVER-STORE

INC FEED 1 ... 10 000

TOOL OFFSET | SETTING DATA | DATA IN-OUT | PART-PROGRAM | DIAG-NOSIS

REF-POINT

E   EMCO specific function (does not exist on SINUMERIK 810T)

function is not realized
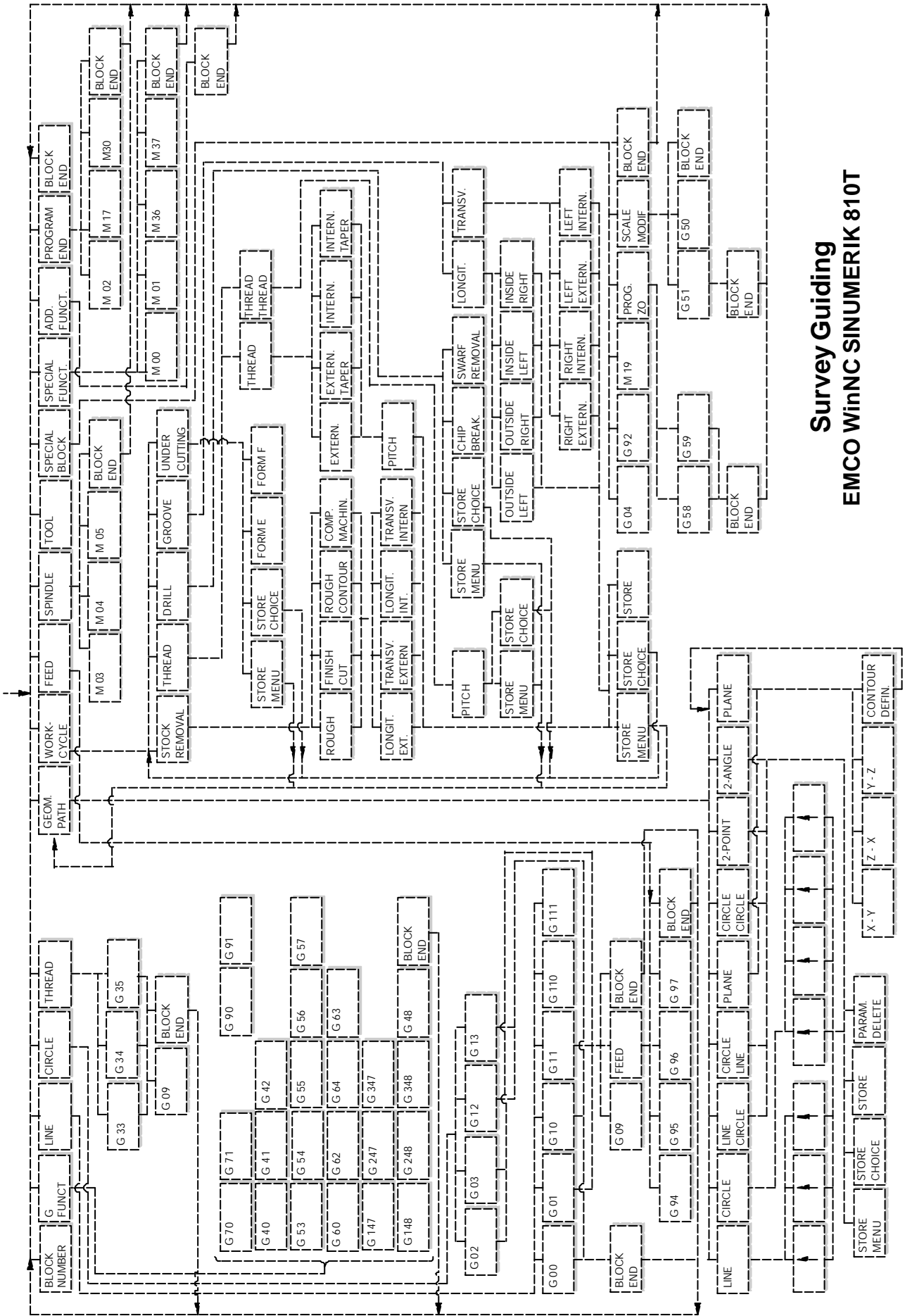
EMCO

**Survey Guiding
EMCO WinNC SINUMERIK 810T**

emco

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 cross section, 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: Block search 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

EMCO

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

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

EMCO

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) Control keyboard

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

Cause: internal error

Remedy: 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

Remedy: 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 control keyboard 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: report to EMCO if repeatable

2550 ORDxx PLC simulation error

Remedy: report to EMCO if repeatable

2551 ORDxx PLC simulation error

Remedy: report to EMCO if repeatable

2562 Read error on CNC program

Cause: error of program file, disk or hard disk error

Remedy: solve problem on DOS level, eventually reinstall software

2614 Nxxxx Internal error MSD

Remedy: report to EMCO if repeatable

2650 Nxxxx Wrong Cycle-Call

Cause: invalid cycle call, if cycle will be called with Gcode

Remedy: report to EMCO if repeatable

2849 Nxxxx Internal error CRC

Remedy: report to EMCO if repeatable

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 geometry 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

EMCO

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 cutter radius

4102 Nxxxx Cutter radius too large

Remedy: correct cutter radius

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.

EMCO

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.

emco

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 !
data I/O

9011 No files found !
data I/O

9014 File already exists !
data I/O

9015 Error while opening file !
data I/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 !
data I/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 TURN 50/55 and the PC TURN 120/125/155.
The alarms 6000 - 6999 normally must be acknowledged with RESET. The alarms 7000 - 7999 are messages which will disappear usually when the alarm releasing situation is eliminated.

PC TURN 50/55
The following alarms are valid for the PC TURN 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 6010.

6012: Z-AXIS NOT READY
see 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 alarm 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 6041.

6043: TOOL CHANGE TIMEOUT
see 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 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 alarm 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 alarm the dividing head is not in a lock position. NC-Start is locked.

PC TURN 120/125/155
The following alarms are valid for the
PC TURN 120/125/155.

6000: EMERGENCY OFF
The EMERGENCY OFF key was pressed.
The reference position will be lost, the auxiliary drives will be switched off.
Remove the endangering situation and restart machine and software.

6001: PLC-CYCLE TIME EXCEEDING
The auxiliary drives will be switched off.
Contact EMCO Service.

6002: PLC - NO PROGRAM CHARGED
The auxiliary drives will be switched off.
Contact EMCO Service.

6003: PLC - NO DATA UNIT
The auxiliary drives will be switched off.
Contact EMCO Service.

6004: PLC - RAM MEMORY FAILURE
The auxiliary drives will be switched off.
Contact EMCO Service.

6008: MISSING CAN SUBSCRIBER
Check fuses or 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, over- or undervoltage from mains.
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 6010.

6012: DRIVE Z-AXIS NOT READY
see 6010.

6013: MAIN DRIVE NOT READY
Main drive power supply defective or main drive too hot, fuse defective, over- or undervoltage from mains.
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 alarm 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 auxiliary drives will be switched off.

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

6040: TOOL TURRET INDEX FAILURE
The tool turret is in no locked position, tool turret sensor board defective, cabling defective, fuse defective.
A running CNC program will be stopped.
Swivel the tool turret with the tool turret key, check fuses or contact EMCO service.

6041: TOOL CHANGE TIMEOUT
Tool drum stucks (collision?), fuse defective, hardware defective.
A running CNC program will be stopped.
Check for collisions, check fuses or contact EMCO service.

6043: TOOL CHANGE TIMEOUT
Tool drum stucks (collision?), fuse defective, hardware defective.
A running CNC program will be stopped.
Check for collisions, check fuses or contact EMCO service.

6046: TOOL TURRET ENCODER FAULT
Fuse defective, hardware defective.
Check fuses or contact EMCO service.

6048: CHUCK NOT READY
Attempt to start the spindle with open chuck or without clamped workpiece.
Chuck stucks mechanically, insufficient pressured air supply, fuse defective, hardware defective.
Check fuses or contact EMCO service.

6049: COLLET NOT READY
see 6048

6050: M25 DURING SPINDLE ROTATION
With M25 the main spindle must stand still (consider run-out time, evtl. program a dwell)

6055: NO PART CLAMPED
This alarm occurs when with rotating spindle the clamping device or the tailstock reach the end position. The workpiece has been pushed out of the chuck or has been pushed into the chuck by the tailstock. Check clamping device settings, clamping forces, alter cutting data.

6056: QUILL NOT READY
Attempt to start the spindle or to move an axis or to swivel the tool turret with undefined tailstock position. Tailstock is locked mechanically (collision), insufficient pressured air supply, fuse defective, magnetic switch defective.
Check for collisions, check fuses or contact EMCO service.

6057: M20/M21 DURING SPINDLE ROTATION
With M20/M21 the main spindle must stand still (consider run-out time, evtl. program a dwell)

6058: M25/M26 DURING QUILL FORWARD
To actuate the clamping device in an NC program with M25 or M26 the tailstock must be in back end position.

6064: AUTOMATIC DOOR NOT READY
Door stucks mechanically (collision), insufficient pressured air supply, limit switch defective, fuse defective.
Check for collisions, check fuses or contact EMCO service.

7000: INVALID TOOL NUMBER PROGRAMMED
The tool position was programmed larger than 8. 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 (also a lubricating pulse will be released).

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 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: TOOL TURRET NOT LOCKED
The tool turret operating was interrupted.
NC start and spindle start are locked. 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 alarm 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.

EMCO

7048: CHUCK OPEN
This message shows that the chuck is open. It will disappear if a workpiece will be clamped.

7049: CHUCK - NO PART CLAMPED
No part is clamped, the spindle can not be switched on.

7050: COLLET OPEN
This message shows that the collet is open. It will disappear if a workpiece will be clamped.

7051: COLLET - NO PART CLAMPED
No part is clamped, the spindle can not be switched on.

7052: QUILL IN UNDEFINED POSITION
The tailstock is in no defined position.
All axis movements, the spindle and the tool turret are locked.
Drive the tailstock in back end position or clamp a workpiece with the tailstock.

7053: QUILL - NO PART CLAMPED
The tailstock reached the front end position. Traverse the tailstock back to the back end position to continue.

EMCO