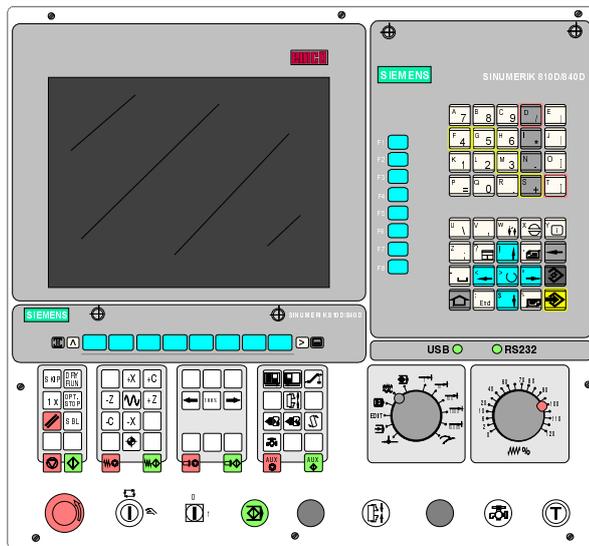# EMCO WinNC SINUMERIK 810D/840D Turning

Software Description/ Software version from 21.00

Software Description
EMCO WinNC
SINUMERIK 810D/840D Turning

Ref.No. EN 1815   Edition G2007-06

This manual is electronically available (.pdf) upon request
at any time on the EMCO homepage.

emco group
Designed for your Profit

Notice

This software description contains all functions that may be carried out with WinNC. However, the availability of functions is dependent on the machine you operate with WinNC.

EMCO

Preface

The EMCO WinNC SINUMERIK 810D/840D Turning Software is part of the EMCO training concept on PC-basis.

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

The milling machines of the EMCO PC 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.

Apart of this software description and the machine description a teaching software CD-ROM "WinTutorial" (CNC examples, operation, description of instructions and cycles) is in preparation.

This manual does not include the whole functionality of the control software SINUMERIK 810D/840D 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 for technical documentation
A-5400 Hallein, Austria

EMCO

Contents

EMCO

EMCO

H: Alarms and Messages H1

Starting Information
see attachment

I: Control Alarms I1

EMCO

A: 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.
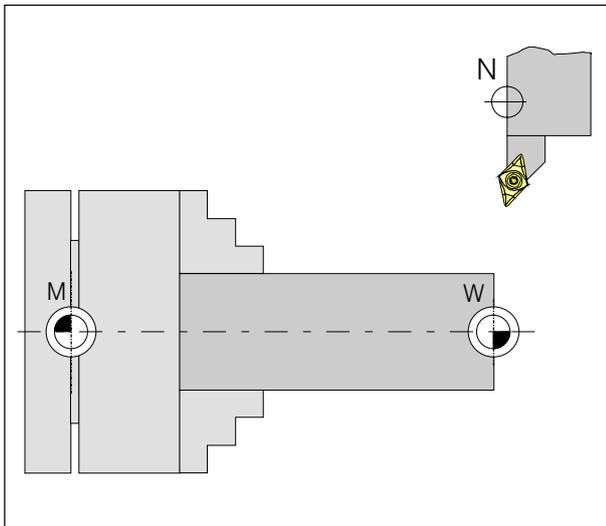
Reference points in the working area

EMCO

Zero Offset

For EMCO lathes the machine zero point "M" is on the turning axis on the 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 Operating Area Parameter - Zero Offsets 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 in any number.
More informations see in the command description.
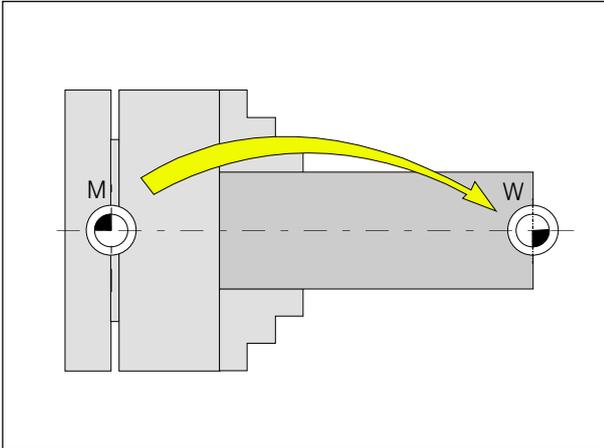
Zero offset from machine zero point M to workpiece zero point W

Coordinate System

The X coordinate is in direction of the cross slide, the Z coordinate in direction of the longitudinal slide.
Koordinatenangaben in Minusrichtung beschreiben Bewegungen des Werkzeugsystems zum Werkstück, Angaben in Plusrichtung vom Werkstück weg.

Coordinate System with Absolute Programming
The origin of the coordinate systemlies in the machine zero point "M" or after a zero offset in the work piece zero point "W".
All target points are described from the origin of the coordinate system by indication of the respective X and Z distances.
X dimensions are programmed as diameter values (like dimensioning on the drawings).

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

Absolute coordinates refer to a fixed point, incremental coordinates to the tool position.
The directions in brackest for X, -X are valid for the PC TURN 50/55, because on thiese machines the tool is in front of the turning axis.

EMCO

Type 500
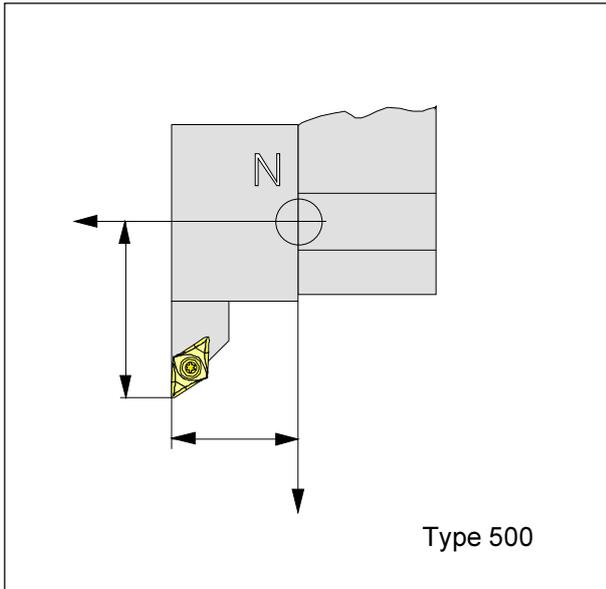
Directions of the length correction of the tool types
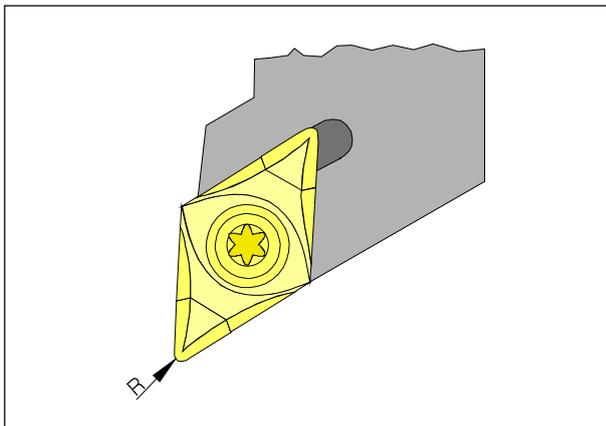
Cutter radius R
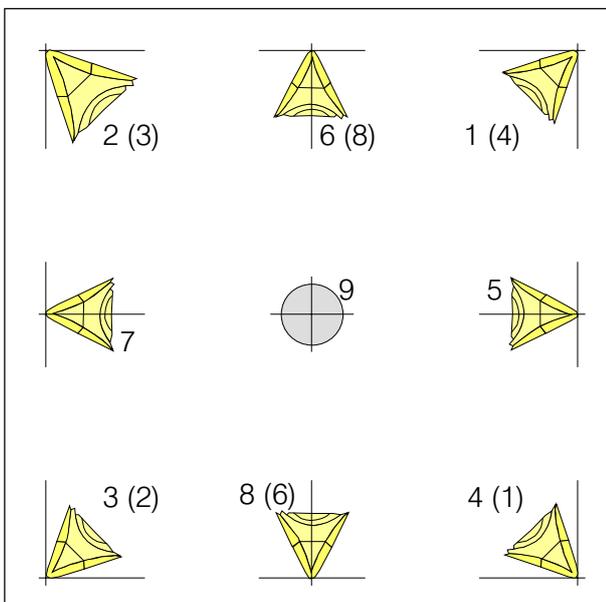
Cutter position of tools

Tool Data

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

Every tool used for machining must be measured. Important is to measure the distance from the tool tip to the tool mount reference point "N".

In the so-called tool data register the measured tool length data, tool position and tool radii can be stored.

The length corrections can be measured half-automatically, the tool position and tool radius must be entered manually
The tool position must be entered always!
Indicating the cutter radius is necessary **only** when a **cutter radius compensation** is used for this tool!

Tool data measuring occurs for Type 500 for:

L1:   in X direction absolute from point "N" in radius
L2:   in Z direction absolute from point "N"
R: cutter radius
Tool type: cutter position (1-9)

Cutter position (tool type)

To determine the tool type look at the tool as it is clamped on the machine.
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 change of the +X direction.

A 3

Type 100

Tool data measuring occurs for Type 100 / 200 for:
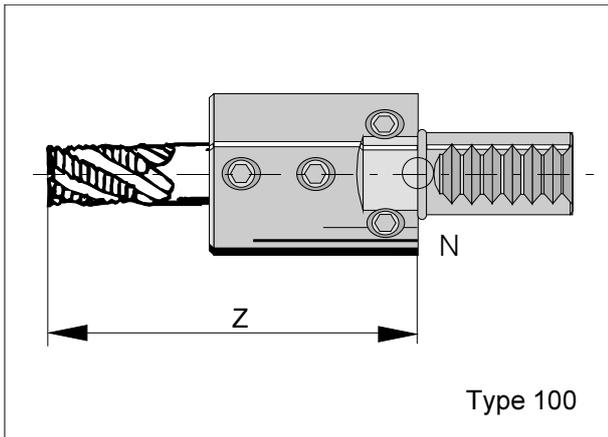
Effect	
G17	Länge 1 in Z
	Länge 3 in X
G18	Länge 2 in X
	Länge 3 in Z
G19	Länge 1 in X
	Länge 2 in Z

Type 100

Type 200

emco
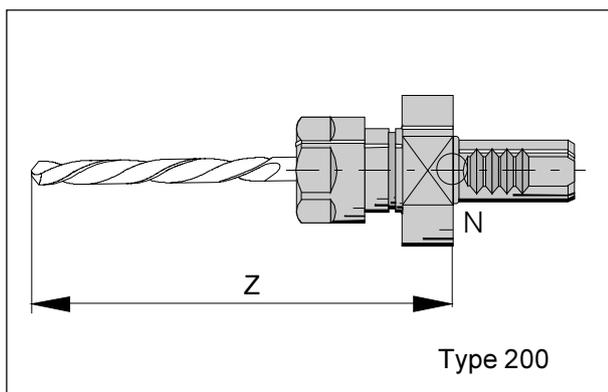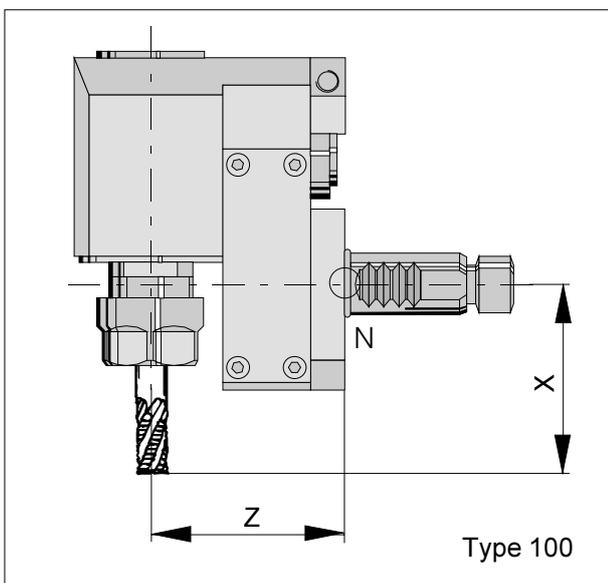
B: Key Description

Control Keyboard, Digitizer Overlay

emco
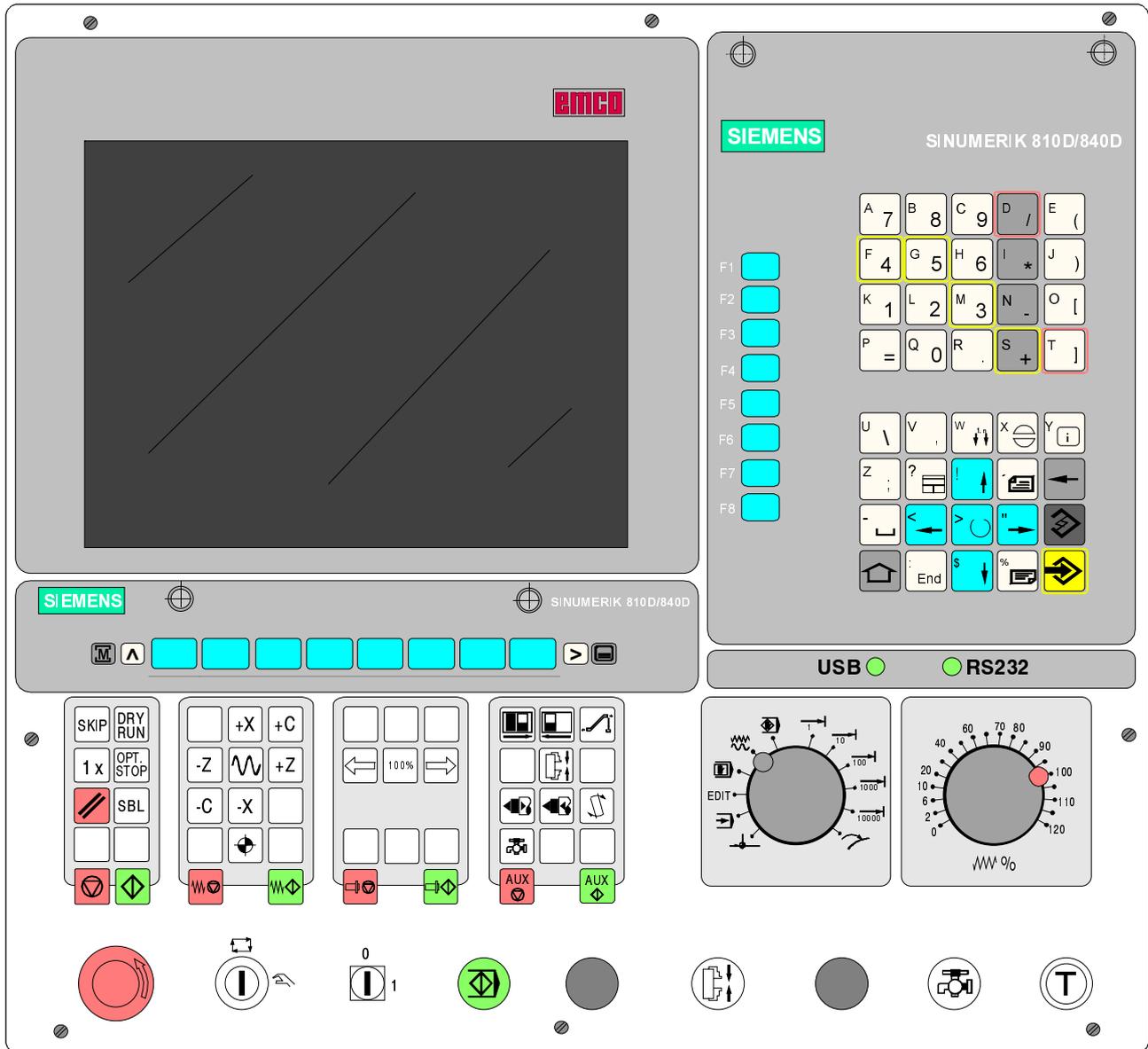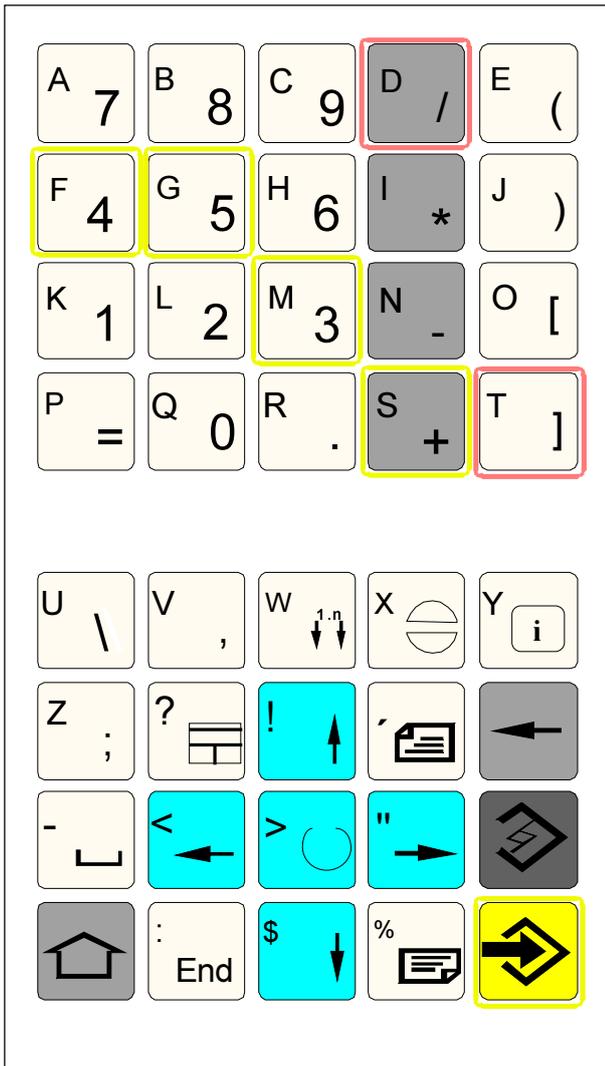
Address and numeric keyboard

Address and Numeric Keyboard

The shift key bottom left shifts to the second key function (indicated in the left top edge of the keys).

Example:

      Leaf backward

      Comma

Double-Shift Function

1 x Shift:
For the following key press the second key function will be done, for all following inputs the first key function.

2 x Shift:
For all following key presses the second key function will be done (shift lock).

3 x Shift:
For the following key press the first key function will be done, for all following inputs the second key function.

4 x Shift:
Deselect the 2x or 3x shift function.

emco

Key Functions

M   Direct jump to the Operating Area Machine

∧   Jump back to the superior menu (recall)

>   Expanding the softkey line in the same menu

Show basic menu (selection Operating Areas)

If pressed again jump back to the previous menu

X   Confirm alarm

Y   Show information for the actual operating status - works only when the dialogue line shows an "*i*".

?   Select window (when several windows are on the screen)

Keyboard inputs are valid for the selected window only.
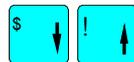
$ ↓   ! ↑   Cursor down / up
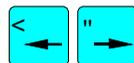
< ←   " →   Cursor left / right

´   %   Leaf backward / forward

- ␣   Blank

←   Clear (Backspace)

> ○   Selection key / Toggle key

- Selection of predefined input values in input fields and lists, which are marked with this symbol
- Activate / disactivate switch box / radio button
  - ⊠ ● = active
  - ☐ ○ = not active

Edit key / Undo

- Switch to edit mode in tables and input fields
- Undo function for table elements and input fields (leaving a filed with this key does not store the entered value but reestablishes the old value)

End   Jump to line end (list end)

Input key

- Take over an edited value
- Open / close directory
- Open file

⌂   Shift key

EMCO

Screen Division

1   Display of the active Operating Area

2   Display of the active channel

3   Operating mode, when a submode is active, it also will be displayed (e.g. REF, INC)

4   Program path and name of the selected program

5   Channel status

6   Channel operating messages

7   Program status

8   Channel status display (SKIP, DRY, SBL, ...)

9   Alarm and message line

10   Working window, NC display
     The working windows (program editor) and NC displays (feed, tool) available in the active Operating Area are displayed here.

11   The selected window is marked with a border and the headline is displayed inverted.
     The keyboard inputs are effective here.

12   Vertical softkeys
     These 8 fields show the functions of the keys right beside. (at the PC: Shift F1..F8)

13   When this symbol is displayed, the key ∧ is active (jump back to superior menu is possible).

14   Dialogue line with operator notes

15   When this symbol is displayed, the key Y/i is active (information available).

16   Horizontal softkeys
     These 8 fields show the functions of the keys below. (at the PC: F1..F8)

17   When this symbol is displayed, the key > is active (more softkey functions available in this line).

PC Keyboard
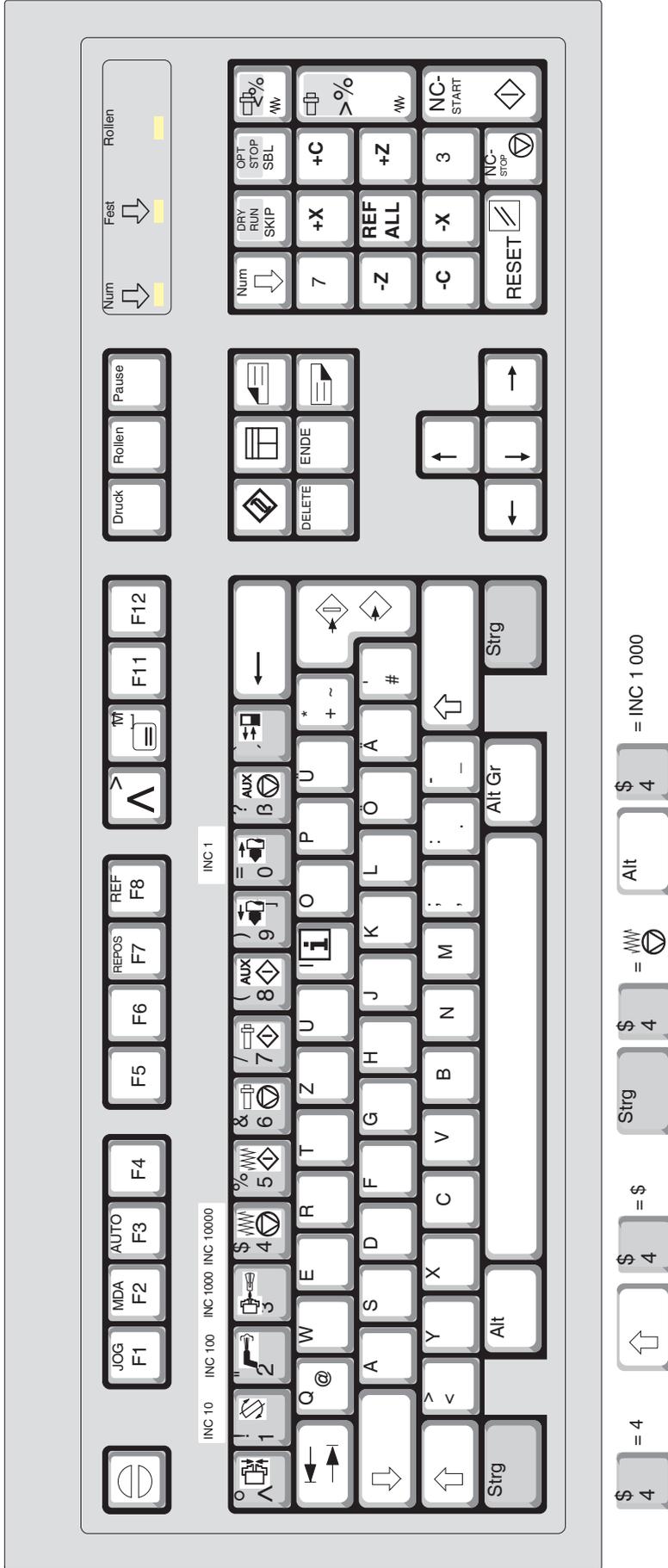
Pressing F10 shows the Operating Areas (Machine, Parameter, ...) in the horizontal softkey line.
Pressing Shift F10 shows the operating modes (AUTOMATIC, JOG, ...) in the vertical softkey line.

Pressing ESC confirms some alarms.

The meaning of the key combination CTRL 2 depends on the machine:
TURN 50/55:   Puff blowing ON/OFF
TURN 100/125/155:   Coolant ON/OFF
The assignement of the accessory functions is described in the chapter "Accessory Functions"

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

emco

Machine control panel
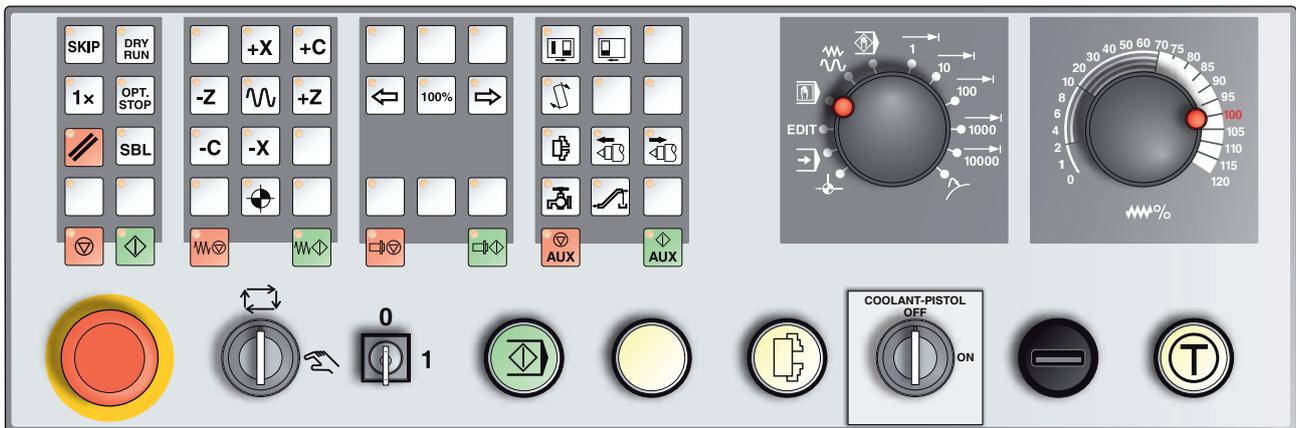
Depending on machine configuration, the control panel can differ slightly from what is shown here.
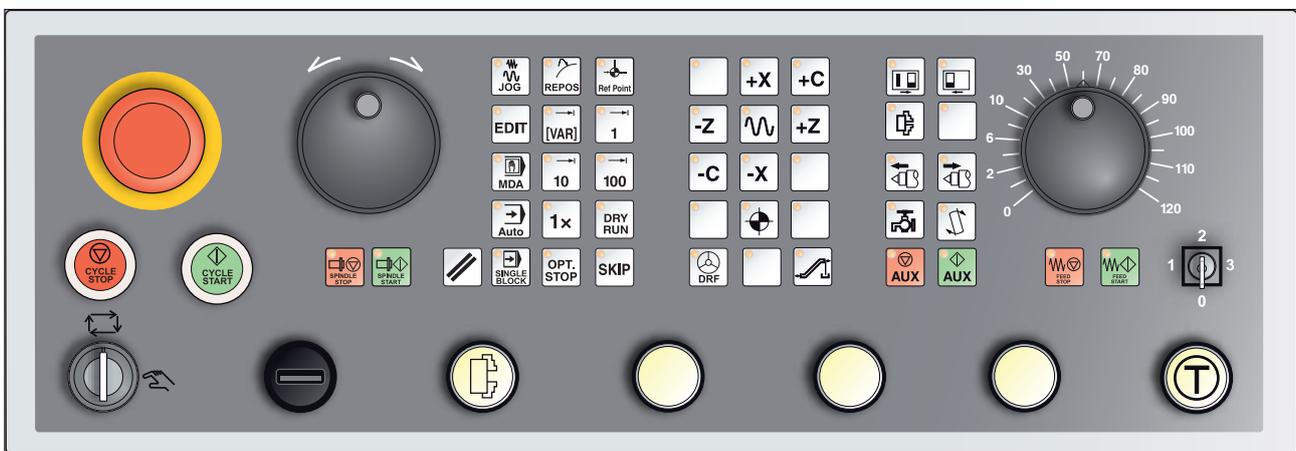
Machine control panel variant with Easy2Control and MOC-Touch

Key description

Instructions:
The buttons for the Concept Turn 250 machine are explained below. For other machines take note of Chapter D EMCO-specific Programming and Operation in the operating manual.

Skip (block mask)

In Skip mode, program blocks will be jumped over when the program is running.

Dryrun (test-run feed)

In Dry-run mode, positioning movements will be carried out with the dry-run feed.

The dry-run feed works instead of the programmed movement commands. On starting the NC program, the main spindle will not be activated and the slides will move with dry-run feed speed.

Only perform the test run without a workpiece to avoid the danger of collisions.

If the test run is engaged, the test "DRY" appears in the simulation window.

Individual piece mode

1× This button makes individual piece mode or constant operation in conjunction with automatic loading equipment available for selection.
Individual state is the default state when switched on.

Optional stop

OPT. STOP When this function is active, program processing will stop at blocks in which the M01 special function has been programmed.
You start processing again with the NC Start button.
If the function is not active, the M01 special function will be ignored.

Reset key (Reset)

• A running program or a movement will be broken off.
• Alarm notifications will be cleared.
• The control system is in the starting position and ready for a new program sequence.

Single block

SBL This functions allows you to execute a program block by block.
The Single Block function can be activated in the automatic mode (a program will be executed automatically) operation type.

When single block processing is active:
• SBL" (=SingleBlock) is shown on the screen.
• the current block of the part programis only processed when you press the NC Start button.
• processing stops after a block is executed.
• the following block is executed by pressing the NC Start key again.
The function can be deselected by pressing the Single Block key again.

NC Stop

After pressing the NC Stop button the execution of the running program will be broken off.
Processing can then be continued by pressing the NC Start button.

NC Start

After pressing the NC Start key the selected program will be started with the current block.

EMCO
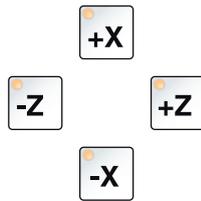
Arrow keys

With these buttons, the NC axes can be moved in JOG operation mode.

Reference point

Pressing this button causes the reference points to be approached in all axes.

Rapid Traverse

If this function is pressed in addition to the movement buttons, the axes concerned move in rapid traverse.

Feed Stop

In "AUTOMATIC" operation mode, this function cancels a slide movement.

Feed Start

This function resumes a programmed slide movement which has been interrupted.
If the main spindle motion was also broken off, it must be switched on first.

Spindle speed correction

The set spindle speed value S will be shown on the screen as an absolute value and as a percentage.
Effective for the main spindle or for the spindle for the driven tools (if present).

Adjustment range:	50 - 120% of the programmed spindle speed
Increment:	5% per button press
100% spindle speed:	100% button

Spindle Stop

This function interrupts the motion of the main spindle. If it happens during a feed movement, that has to be stopped first.

Spindle Start

This function resumes the programmed spindle motion.

Automatic machine doors

To open and close the machine doors.

EMCO

Chip conveyor (Option)

Switch on chip conveyor:
Forwards: Press button for less than 1 second.
Backwards: Press button for longer than 1 second.

The chip conveyor will be switched off after a defined time (approx. 35 seconds).
This value is set in the factory.

Clamping device

This button activates the clamping device. For chuck/pliers changeover see "Machine configuration".

Via NC program
M25   Open clamping device
M26   Close clamping device

Manual tool change

Pressing this button starts a manual tool change.
The clamped tool can now be removed and replaced.

Preconditions:
• Machine doors closed
• "JOG" operating mode
• Key switch in "Hand" position

> **Information:**
> • Interrupt the change process by moving the override switch below 4%.
> • Cancellation of the change procedure by pressing the reset button.

Tailstock forwards, tailstock backwards

These buttons move the tailstock forwards or backwards.

Coolant

This function switches the coolant equipment on or off.

Auxiliary OFF

This function switches off the machine's auxiliary unit. Only effective if spindle and program are off.

Auxiliary ON

This function makes the machine's auxiliary unit ready for operation (e.g.: hydraulics, feed drives, spindle drives, lubrication, chip conveyors, coolant).
The button must be pressed for around 1 second.
Briefly pressing the AUX ON button is a quit function and causes the central lubrication system to perform a lubrication impulse.

EMCO
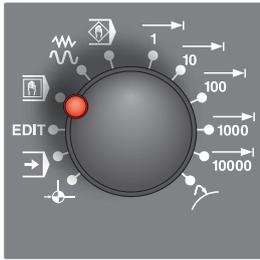
Types of operation

REF - Reference mode
Approaching the reference point (Ref) in the JOG operating mode.

AUTO - Automatic mode
Control the machine by automatically executing programs.
Here part programs are selected, started, adjusted, deliberately influenced (e.g. individual block) and executed.

EDIT
no function

MDA - Semi-automatic mode
Control the machine by executing a block or a sequence of blocks. Block input is performed via the operating panel.

JOG - Jogging
Standard movement of the machine by continuous movement of the axes via the directional buttons or by incremental movement of the axes via the directional buttons or the handwheel.
JOG is used in manual mode as well as for set-up of the machine.

TEACH IN
no function

Inc 1 - Incremental feed
Move step by step a predefined distance of 1 increment.
Metrical measurement system: Inc 1 corresponds to 1µm
Imperial (inch-based) measurement system: Inc 1 equates to 0.1 µinch

Inc 10 - Incremental feed
Move step by step a predefined distance of 10 increments.
Metrical measurement system: Inc 10 corresponds to 10µm
Imperial (inch-based) measurement system: Inc 10 equates to 1 µinch

Inc 100 - Incremental feed
Move step by step a predefined distance of 100 increments.
Metrical measurement system: Inc 100 corresponds to 100µm
Imperial (inch-based) measurement system: Inc 100 equates to 10 µinch

EMCO

Inc 1000 - Incremental feed
Move step by step a predefined distance of 1000 increments.
Metrical measurement system: Inc 1000 corresponds to 1000µm
Imperial (inch-based) measurement system: Inc 1000 equates to 100 µinch
(equates to 1 mm)

Inc 10000 - Incremental feed
 • Move step by step a predefined distance of 10,000 increments.
Metrical measurement system: Inc 10000 corresponds to 10000µm
Imperial (inch-based) measurement system: Inc 10000 equates to 1000 µinch
(equates to 10 mm)
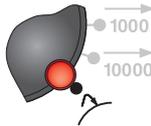
REPOS - Repositioning
Back-positioning, approach contour again in the JOG operating mode

Information:
 • The operating modes can be selected via softkeys (PC keyboard) or with the operating mode selector switch.
 • Switching between the metrical measurement system and the imperial (inch-based) measurement system is carried out with the EmConfig utility software (see Chapter X EmConfig).

Instructions:
Metric is assigned in the imperial measuring system as follows:

Feed:
Millimeters in inches:
mm/min => inches/min
mm/U => inches/U

Constant cutting speed:
Meters in feet:
m/min => feet/min

emco
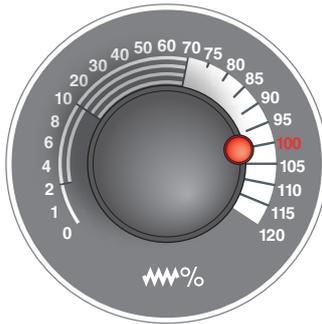
Override switch (feed rate override)

The rotary switch with notch positions changes the programmed feed value F (corresponds to 100%).
The set feed value F in % will be shown on the screen.

Adjustment range:
0% to 120% of the programmed feed.
In rapid traverse 100% will not be exceeded.

No effect with thread commands G33, G63
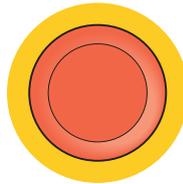
EMERGENCY STOP

Press the red button only in an emergency.

Effects:
As a rule, the EMERGENCY STOP button will lead to all drives being stopped with the greatest possible braking torque.

To continue working, press the following buttons:
RESET, AUX ON, doors OPEN and CLOSED.
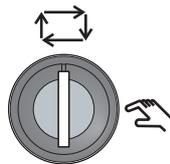
Key Switch Special Operations Mode

The key switch can be switched to the "AUTOMATIC" or "SET-UP" (hand) position.
With this key switch it is possible to perform movements in Jog Mode when the sliding door is open.

Danger:

When Special Operations mode is active, the danger of accidents is increased.
The key for this switch should only be held by persons who have the required knowledge about the dangers and exercise appropriate care.
Keep the chip guard door closed even in Set-up mode.
Keys should only be used by authorized persons.
After work is carried out in Special Operations mode, always withdraw key (accident danger).
Observe country-specific safety instructions (e.g.: SUVA, BG, UVV ....).

emco

Multifunction switch for operating modes
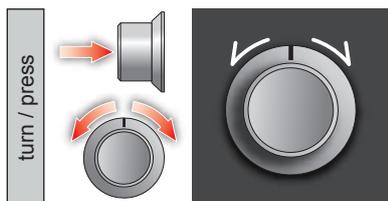
The multi-function switch is designed as a rotary switch with a press feature.

Populated function
- The user interface is opened by pressing the multifunction operation. The active function is indicated by a green check-box.
- Turning the switch allows you to switch between the functions. The black bar with the symbols moves to the left or to the right.
- Activating a function or a change to a sub-menu is executed by pressing the button.

The interface offers the following functions:

Overwiew

1 Spindle override: controls the spindle speed equivalent to conventional spindle override

2 Feed override: controls the feed rate equivalent to conventional feed override

3 Modes: allows you to select the operating mode using the multifunction operation

4 Close: The user interface is closed. The menu disappears, return to the control surface

5 Settings: opens another level with settings

6 Cursor: shows the actual position in the menu

Note:
The functionality of the multifunction operation is depending on the installed software version.

EMCO
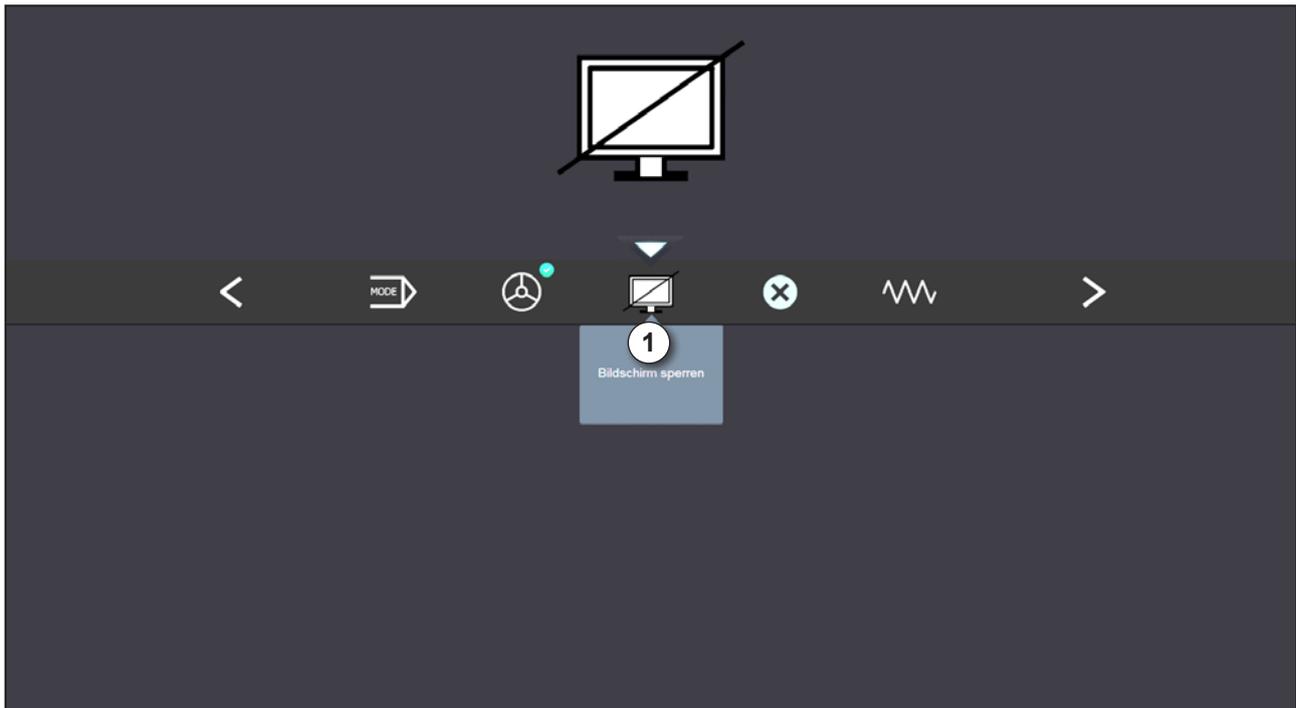
Lock screen

1 Lock screen: pressing again unlocks the screen and closes the unser interface.
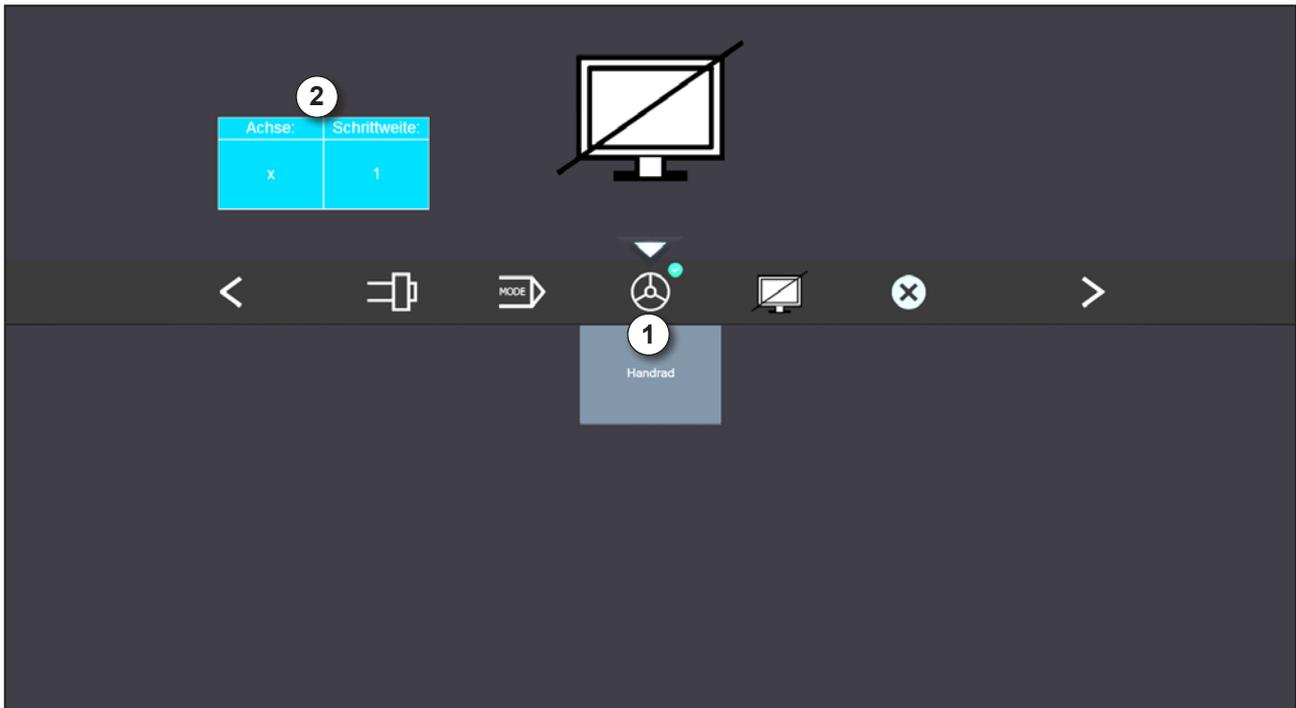
Handwheel function

The handwheel (1) activates the handwheel mode. The parameters for axis and step width (2) are set with the axis- and operating -mode buttons on the machine keyboard.

Operation

- The electronic handwheel is used to traverse the slides at a defined step width.
- The step width depends on the selected Inc mode: Inc 1, Inc 10, Inc 100.

- There must be one pre-selected Inc mode and an axis defined by a direction key.

- Also refer to "types of operation" und "arrow keys" in chapter B.

Note:
In the mode "Inc 1000" the slides cannot be moved with the handwheel. "Inc 1000" operates with "Inc 100".

Key switch

The key switch function is machine-specific.

Additional NC start button

The additional button has the same function as on the machine control panel.
(Double movement because of better operation).

USB connection (USB 2.0)

Data is exchanged with the machine (data copying, software installation) via this USB connection.

Enable button

When the door is open, axis movements via direction buttons and tool changer movements are authorized by pressing the enable button (pre-condition key switch in SET-UP position).
In machines with automated doors (option) pressing the enable switch opens the machine doors.

EMCO

C: Operation

Operation principle

The operation of the SINUMERIK 810D/840D is organized in 6 menus, so-called Operating Areas:

- Machine
- Parameter
- Program
- Services
- Diagnosis
- Start-Up

These six operating areas are displayed in the basic menu in the horizontal softkey line.

Call basic menu

Press the key [image] to display the basic menu with the six operating areas in the horizontal softkey line.
From any menu the basic menu can be called with this key.
By pressing again this key you will go back to the previous menu.

Navigation in the menu window

- Change menu window
  With this key you can change the active window (the active window is marked with a coloured border).
  Inputs can be done in the active window only.

- Scrolling in the menu window
  Scrolls page forward or back.

- Place cursor in the menu window.

EMCO
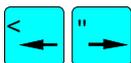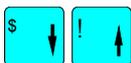
Navigation in the directories
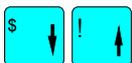
- Select file / directory

- Open / close directory
  Open / close selected directory.

- Open file
  Open file when it should be processed in the editor..

Edit inputs / values

- Use the key to activate / disactivate radiobuttons or switchboxes.
  ⊠ ● = active
  ☐ ○ = not active

- Input fields

  Change into input mode.

  Enter a value or term (e.g. file name) with the alphanumeric keyboard.

  You automatically enter the input mode, when the cursor was placed on the input field previously.

  Acknowledge your input with the key "Input". The value will be taken over.

  Use the key to toggle between predefined values (e.g. forward-backward).

Confirm / cancel input

OK

- Confirm input
  Save inputs and leave actual menu window (return to calling menu).

CANCEL

- Cancel input
  Reject inputs and leave actual menu window (return to calling menu).

Λ

Reject inputs and leave actual menu window (return automatically to the next higher menu level).

Reject inputs and stay in actual menu window.

Mouse operation

- 1x Click means:
  Activate menu window
  Put cursor on desired input field
  Select directory
  Press softkey
  Activate / disactivate radio button / switchbox
  Activate input field
  Open selection lis

- 2x Click (double click) means:
  List selection
  Take over value / input
  Open directory

- Right mouse key means

  Show operating areas

EMCO

Survey Operating Areas

The functionalities of the control are organized in operating areas.

Operating Area	executable functions
Machine	Work off part program Manual operation of the machine
Parameter	Editing data for programs and tool administration
Program	Creating and adapting part programs
Services	Read in / out programs and data
Diagnosis	Alarm display Service display
Start-Up	Adjusting the NC data to the machineSystem settings

EMCO

Operating Area Machine

The Operating Area Maschine covers all functions and influences, that lead to actions at the machine tool or detect its state.

Three operation modes:

- JOG

  Jog is used for manual operation and set-up of the machine.
  Set-up functions::

  Approach reference point (Ref)

  Repositioning (Repos)

  Increment traverse

- MDA

  half-automatic operation
  Part programs can be created and worked off block-by-block.

- AUTOMATIC

  Fully automatic operation
  Working off part programs.
  Part programs will be selected, started, corrected, intentionally influenced (e.g. single block) and worked off.

These operating modes can be selected by softkey (PC keyboard) or with the operating mode selector switch.

EMCO

Approach reference point

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

- Select REF mode (or Alt+F8 at the PC).

- Press the direction key -X or +X to approach the reference point in this axis, same for all other axes.

- With the key REF ALL all axes will be approached automatically (PC keyboard).

Danger of collisions

Mind for obstacles in the working area (clamping devices, clamped workpieces etc.).

After reaching the reference point its position will be displayed as actual position. Now the control is synchronized to the machine.

Traverse slides manually

The machine axes can be traversed manually with the direction keys.

- Change to JOG mode (or Alt+F1 at the PC).

- The keys -X , +X , -Y , +Y , -Z , +Z , etc. move the axes in the desired direction as long as the keys are pressed.

- Feed will be set with the override switch.

- When the key is pressed simultanuously, the axes move with rapid speed.
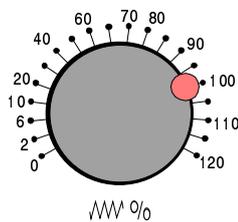
Traverse slides in increments

With the direction keys you can traverse the slides in increments.

INC 1	1/1000 mm	per key press
INC 10	1/100 mm	per key press
INC 100	1/10 mm	per key press
INC 1000	1 mm	per key press
INC VAR	variable step measure	

- Change into INC mode (⟶|₁ ... ⟶|₁₀₀₀₀ or Alt+0 ... Alt+4 at the PC).
  (Alt 0..10^0=1, Alt 1..10^1=10, Alt 2..10^2=100, ...)

- The keys -X , +X , -Y , +Y , -Z , +Z , etc. move the axes in the desired direction for the desired increment.

- Feed will be set with the override switch.

- When the key ⟋⋁⟍ is pressed simultanuously, the axes move with rapid speed.

EMCO
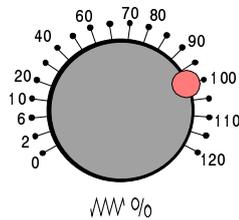
MDA mode

In the MDA mode (Manual Data Automatic) part programs can be created and ran block-by-block. Therefore enter the desired movements via the keyboard as single part program blocks into the control.

The control works off the blocks after pressing the key ⬨.

For a MDA program run the same preconditions are valid as for an automatic program run.

Automatic mode

In the Automatic mode part programs can run fully automatic.

Preconditions for working off part programs:
• The reference point was approached
• The part program is loaded into the control.
• The necessary correction values have been checked or entered (e.g. zero offsets, tool corrections)
• The safety locks are active (e.g. chip guard door closed).

Possibilities in automatic mode:
• Program correction
• Block search
• Overstore
• Program influence

see chapter F - Program Run.

EMCO

Operating Area Parameter

In the Operating Area Parameter you can enter and edit data for program and tool correction.

Tool data

see chapter E - Tool Measuring / Tool Administration.

R Parameter (arithmetic parameter)

R Parameter are variables which can be used as calculation parameter within programs.
These parameter can be edited manually in this operating range.

Press the softkey R PARAMETER.
You can page up and down in the R parameter list

| R para- |
| meter |

using the keys [image] and [image] .

Change parameter:
Position the cursor on the appropriate input filed and enter the new value.

Delete parameter:

| Delete |
| area |

With the softkey DELETE AREA you can delete all parameters in the range R.. to R.. .

| Delete all |

The softkey DELETE ALL deletes the entire R parameter range.
With the softkeys ABORT and OK you can abort or confirm deleting.

| Abort | | OK |

Find parameter:
Press the softkey SEARCH and enter the number of the parameter to be found.

| Search |

When you press [image] the cursor will be placed on the specified parameter if it exists.

EMCO

100 computation variables (= R parameter) of the REAL type are available as a standard under the address R with the control Sinumerik 840 D.
The range R0 to R89 is at the free disposal of the user, R90 to R99 is reserved for EMCO.

Workpiece counter (R90, R91)

Actual workpiece number
The actual workpiece number is indicated under parameter R90.

Nominal workpiece number
The nominal workpiece number is indicated under parameter R91.

Function
• If the workpiece number is entered in R91, parameter R90 counts from the preset nominal workpiece number downward to 0.
  When the preset workpiece number is worked off, the message "Nominal workpiece number reached" is displayed.
• If the workpiece number R91 and the actual workpiece number R90 is set to 0, parameter R90 counts from 0 upward.

Programming
The call-up of the workpiece counter in the program is carried out just before the M30 command with **L700 P1**.

Example
250 workpieces are manufactured.

• Nominal workpiece number R91 = 250 set
  The workpiece counter counts from 250 to 0 downward and then emits the message "Nominal workpiece number reached".

• Nominal workpiece number R91 = 0 set
  Actual workpiece number R90 = 0 set
  The workpiece counter counts from 0 to 250 and does not emit a message.

Setting data

Work. area
limitation

- Working area limitation

With the working area limitation you can set limits in which the tool can be moved.

Place the cursor on the desired input field and enter the new value.

Activate your input with the key `> ◯`.

Note:
In MDA and Automatic modes, within an NC program the working area limitation is active only after the WALIMON command.

JOG
data

- JOG data

JOG feedrate
Feed for the axes for manual traversin in JOG operation.

Enter the new value

Spindle
data

- Spindle data

Max. / min:
Limitation of the spindle speed. It is permitted only within the maximum and minimum values that are defined in the machine data

Spindle speed limitation at G96:
Programmable upper spindle speed limit (G96) for constant cutting speed.

Gear:
Enter the selected gear step for machines with mechanical gear.
By that the speed of the main spindle can be monitored correctly.

Place the cursor on the input field and enter the new

value or select the value with the key `> ◯`.

EMCO

Feedrate DRY

- Dry run feedrate

This feedrate is used in DRY RUN instead of the programmed feedrate.

Starting angle

- Starting angle for thread cutting

For thread cutting, a starting position for the spindle is displayed as the initial angle. A multiple thread can be cut by changing the angle when the thread cut operation is repeated.

EMCO

Zero offset

• Change settable zero offset (G54 - G57):

Coarse offset:
On most machine tools, this value is secured against unauthorized changes with a key switch.

Fine offset:
Is used for fine corrections of the coarse offset (e.g. wear correction) and is not secured against changes. The input value for fine offset is limited to ± 1 mm.

The effective settable zero offset is the total sum of fine and coarse offset.

Rotation, scale, mirror:
Determination like programming of "Frames". Rotation can only be set around geometry axes.
See chapter D - Programming - Coordinates, Zero offsets.

WO+	WO-
	Selected WO
	Accept position
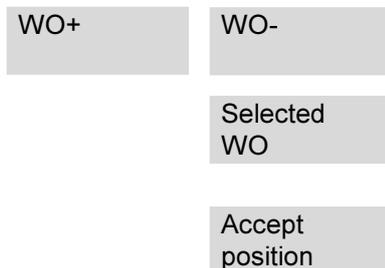
Select the desired zero offset G54 - G57 with the softkeys WO- and WO+.
Select the active zero offset (in MDA or Automatic) with the softkey SELECTED WO.

The softkey ACCEPT POSITION is displayed only if an axis position is entered in the input field. This position is transferred to the control when you press ACCEPT POSITION.

Enter the values into the input field or select a new value with the key ⟩◯ (mirror).

Reject	Save

SAVE will save the new values, REJECT resets the altered values to the original values.

EMCO

• Measure settable zero offset (G54 - G57) (touch):

| ZO+ | ZO- |

Select the desired zero offset G54 - G57 with the softkeys ZO- and ZO+.

| | Selected ZO |

Select the active zero offset (in MDA or Automatic) with the softkey SELECTED ZO.

You must be in JOG mode.

| | Determine ZO |

Press the softkey DETERMINE ZO.

Enter the corresponding tool data for your touching tool:
T-No. tool
D-No. correction (edge)

Use the key [>] to select:

- the relevant length parameter (1, 2, 3)
  and direction (+, -, without)
- inclusion and direction of radius (+, -, without)
- inclusion and direction of a freely definable offset
  (+, -, without)

| | OK |

Confirm the touching tool with OK.

Place the cursor on the corresponding input field of the offset (e.g. Z coarse).

| | OK |

Touch on the corresponding position of the zero offset (e.g. clamping device position in Z) and press the softkey OK.
The position will be taken over.

Touch on all desired positions of the zero offset.

| Reject | Save |

SAVE will save the new values, REJECT resets the altered values to the original values.

EMCO

- Change base zero offset:

The base zero offset is a zero offset which is always active (without special call-up).
The base zero offset is used for e.g. spacer tables on milling machines or intermediate flanges on lathes.

Coarse offset:
On most machine tools, this value is secured against unauthorized changes with a key switch.

Fine offset:
Is used for fine corrections of the coarse offset (e.g. wear correction) and is not secured against changes. The input value for fine offset is limited to ± 1 mm.

The effective base zero offset is the total sum of fine and coarse offset.

Rotation, scale, mirror:
Determination like programming of "Frames". Rotation can only be set around geometry axes.
See chapter D - Programming - Coordinates, Zero offsets.

Overview	Base ZO

Select base zero offset:
Press the softkeys OVERVIEW and BASE ZO.

Input of the baser zero offset is the same like for settable zero offsets.

Total effefctive zero offset

The total zero offset, thath is active in a part program, is the sum of base zero offset + settable zero offset G54-G599 + Frames.

Total ZO = Base + G54-G599 + Frames
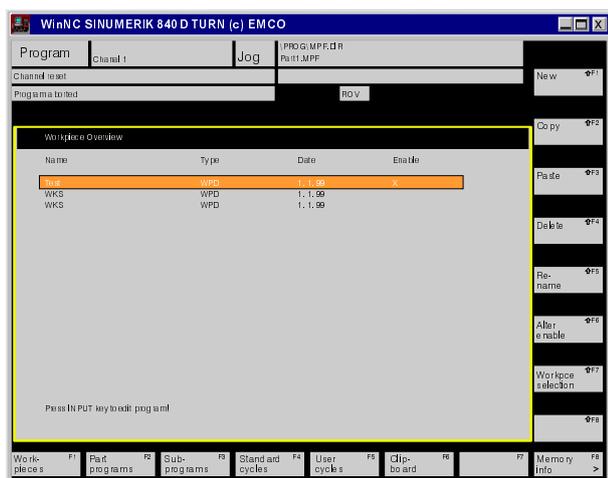
Operating Area Program

In the Operating Area Program, part programs can be created and adapted, and part programs can be administrated.

Program types

- Part program

A part program is a sequence of commands for machining a workpiece.

- Subprogram

A subprogram is a sequence of part program commands, that can be called up multiple with different supply parameters.
Cycles are a kind of subprograms

- Workpiece

In this context a workpiece is a directory, that includes programs or other data.

- Cycles

Cycles are subprograms to execute repeated used machining steps at the workpiece.
The preprogrammed standard cycles can not be altered.
User cycles can be created and altered at will.

Program administration

- File and directory types

name.MPF	main program
name.SPF	subprogram

name.TOA	tool data
name.UFR	zero offset / Frame
name.INI	initializing file
name.COM	comment
name.DEF	definition for global user data and makros

name.DIR	common directory, contains programs, workpiece directories and other directories with the extension .DIR. The names of these directories (MPF.DIR, DPF.DIR, CLIP.DIR, ...) are predefined and can not be altered.
name.WPD	workpiece directory, contains program and data modules, which belong to the workpiece (it must not contain other directories with the extension .DIR or .WPD)
name.CLP	clipboard directory, may contain all types of files and directories.

- Copy / Paste

Place the cursor on the file to be copied and press the softkey COPY.

Copy

The file will be marked as copy source.

Enter the directory, in which the marked file should be copied and press the softkey PASTE.
With paste into a workpiece directory the type can be

Paste

altered with the key [> ○].

When the name of the source file and the copied file should be the same, press the softkey OK.

OK

When the copied file should get a new name, enter the new name with the keyboard and press the softkey OK.

emco

• Rename

Place the cursor on the file to be renamed and press the softkey RENAME.
The rename dialogue window will be opened.

Enter the new name.

The file type can be changed with the key [> ⟳].

Re-name

• Delete

Place the cursor on the file to be deleted.

To mark several files, place the cursor on the first file,

press the key [> ⟳] and place the cursor on the last

file.

Delete

Press the softkey DELETE.
Confirm the safety query with the softkey OK and all marked files / directories will be deleted.

OK

- Programs can be deleted only when they are not in process.
- To delete a workpiece directory, no program in this workpiece directory must be selected.
- When a workpiece directory is deleted, all files within this directory will be deleted.

• Enable

A program can be worked off only when it is enabled. Enabled programs are marked with an "X" in the program list.

Alter enable

To enable or disable a program, mark the program and press the softkey ALTER ENABLE.

emco

Create workpiece directory

Work-
piece

new

Press the softkey WORKPIECE. Die Übersicht aller Werkstückverzeichnisse wird eingeblendet.
Press the softkey NEW. The input window will be opened.
Enter the name for the new workpiece directory with the keyboard.

The name of the first part program will be asked for and the editor for this part program will be opened.

Create / edit program

• Create new program

Open the workpiece directory in which the new program should be created.

new

Press the softkey NEW and enter the new program name.
The corresponding file name can be selected with the

key .

• Select existing program

Place the cursor on the program that should be edited.

Press the key .

The text editor for the selected file will be opened.

EMCO
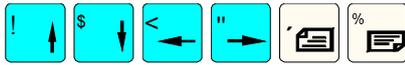
- Edit program

Place the cursor in the text with the cursor keys and the keys page up / page down.

The delete key deletes the character left from the cursor.

The input key closes a block. A LF sign (line feed) will be written into the program. Only after that the entered block will be accepted for machining.

Overwrite	Paste

With the softkeys OVERWRITE and PASTE you can toggle between overwrite and insert mode.

Mark, copy, paste, delete block:

Place the cursor on the beginning of the block and press the softkey MARK BLOCK.
Move the cursor on the end of the block and the block will be marked automatically.

Mark block

COPY BLOCK copies the block in the clipboard. Also with program change the block stays in the clipboard.

Copy block

INSERT BLOCK inserts the block from the clipboard into the text before the cursor position.

Insert block

DELETE BLOCK deletes the marked block.

Delete block

By pressing again MARK BLOCK you will leave the marking mode.

Renumber:

With the softkey RENUMBER the program blocks in the editor will be renumbered.

Renumber

CLOSE
A query will be displayed whether the changes should be stored. The the text editor will be closed and the screen shows the program overview.

Close

EMCO
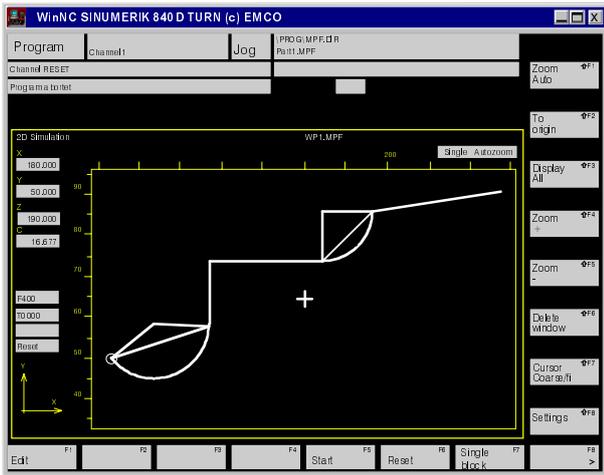
Program simulation

While editing a program the programmed tool movements can be simulated graphically at the screen.
By that the program can be tested for geometrical and formal correctness.
Not recognized will be technological mistakes (e.g. wrong sense of rotation, wrong feedrate, etc.).

Display colors:

light green	traversing path with feed
dark green	traversing path with rapid feed
yellow	reticule, tool symbol, symmetry axes etc.
blue	circle auxiliary lines

The simulation window shows the actual ayis poaitions, feed, tool, status Run/Reset and the settings Autozoom and Single.

Simulation	Press the softkey SIMULATION.
Start	The softkey START starts the simulation.
Reset	RESET sets back the simulation.
Single	With SINGLE the simulation runs block-by-block (continue with START).
Zoom Auto	ZOOM AUTO displays all traversing pathes in the graphic traversing area adapted to the window size.
To Origin	TO ORIGIN reestablishes the basic picture (cancel zoom functions)
Display all	DISPLAY ALL shows the complete traversing area of the machine.
Zoom + / Zoom -	ZOOM + and ZOOM - set the zoom factor of the display. Previously put the reticule with the cursor keys to the desired centre of display.
Delete window	DELETE WINDOW clears the simulation display.
Cursor coarse/fine	CURSOR COARSE/FINE changes the step with of the cursor steps.
Edit	EDIT returns to the program editor.

emco

Simulation settings:

Settings

The softkey SETTINGS opens the settings window for the simulation.

2D simulation setup data

View plane

X: ☑ Z   Y: ☑ X   ☒ Circle with auxiliary lines
                                ☐ Use tool offsets

Draw delay   ☐ Activate cutter

    0   ms

                               ☑ 1   Actual channel

   ┌ Position type ──────┐
   │ ○ MCS position   │
   │ ● WCS Position   │
   └─────────────────────┘
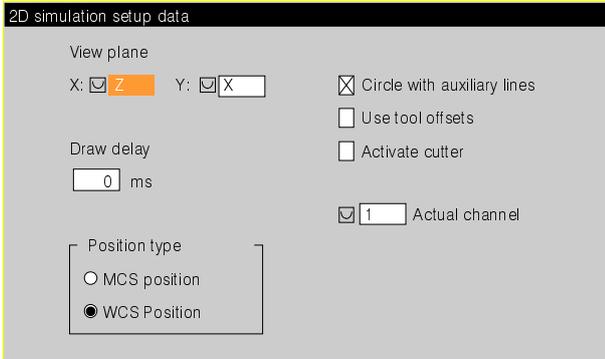
Normal presentation level

2D simulation setup data

View plane

X: ☑ Z   Y: ☑ y-Tr   ☒ Circle with auxiliary lines
                                ☐ Use tool offsets

Draw delay   ☐ Activate cutter

    0   ms

                               ☑ 1   Actual channel

   ┌ Position type ──────┐
   │ ○ MCS position   │
   │ ● WCS Position   │
   └─────────────────────┘

Presentation levels - selection for Tracyl

2D Simulation setup data

View plane

X: ☑ X- Tr   Y: ☑ y-Tr   ☒ Circle with auxiliary lines
                                ☐ Use tool offsets

Draw delay   ☐ Activate cutter

    0   ms

                               ☑ 1   Actual channel

   ┌ Position type ──────┐
   │ ○ MCS Position   │
   │ ● WCS Position   │
   └─────────────────────┘

Presentation levels - selection for Transmit

With view plane you can decide, which plane is displayed in simulation.

Below draw delay you can enter a delay time between the simulation blocks. By that the simulation will be more representative.

When you press the key ⟨X⟩ while a running simulation, the rest of the simulation will run without delay.

Below position type you can select whether the simulation data will be displayed in the machine coordinate system (MCS) or in the workpiece coordinate system (WCS).

Circle with auxiliary lines
Select / deselect the display of radii and the string between the circle end points.

Use tool offsets
Simulation with / without tool offsets.

Activate cutter
Simulation with cutter radius compensation or without cutter radius compensation (display of tool center path).

Actual channel
Selection of the actual channel (only on machines with more than one channel).

The main program, the subroutines and the cycles must be released to enable simulation.

---

emco
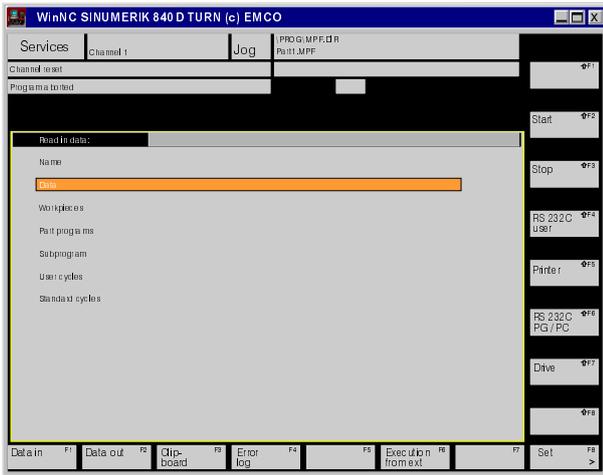
Operating Area Services

The Operating Area Services is used to read in or send data via interface COM1 - COM4.

Print data with the softkey PRINTER, and transmit data to / from disk drive with the softkey DRIVE.
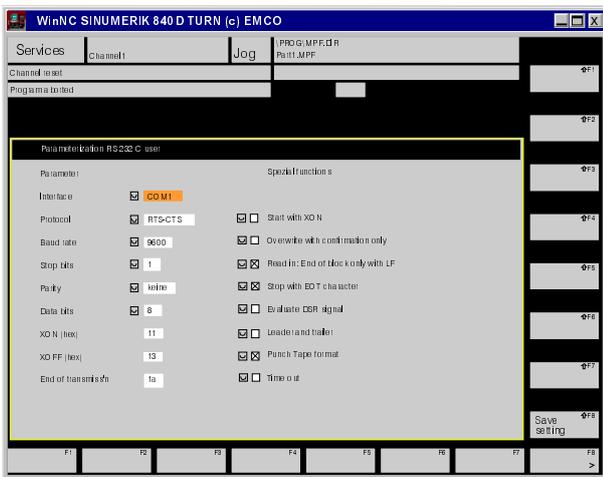
Interface settings

For data transmission the settings of sender and receiver must be the same, otherwise transmission would not work.

With EMCO WinNC you can send only via the interface RS 232 C User.
The function RS 232C PG/PC is not active.

Press the softkeys RS232 C USER and SET.

Place the cursor on the input fields and enter the corresponding values.

The values of the parameter Interface, Protocol, Baud rate, Stop bits, Parity, Data bits can be selected

with the toggle key ( ).

The parameters Protocol, Baud rate, Stopbits, Parity, Data bits, XON-Sign, XOFF-Sign and DSR Signal are getting seperately saved for every interface. While changing the interface the new parameters are shown and can be modified.
Also the special functions can be selected with the toggle key.

The settings can be saved with the softkey SAVE SETTING.

Drive settings

Press the softkeys DRIVE and SET.

Select Floppy (Drive A:) or Free Directory.
With the option Free Directory you can select any directory, e.g. on hard disk C:.

The special functions can be selected with the toggle key.

Save setting

Save setting

The settings can be saved with the softkey SAVE SETTING.

EMCO
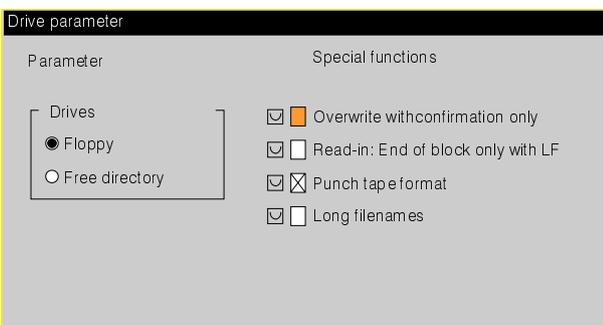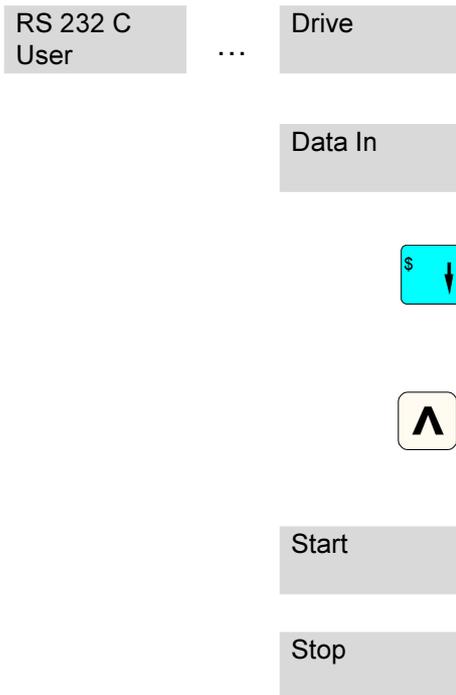
Read-in data

RS 232 C User	…	Drive

Select data source with softkey RS 232 C USER to DRIVE.

Data In

Press the softkey DATA IN.

Place the cursor on the desired directory in the directory list.
The data read-in will be written into this selected directory.

With the softkey ∧ you will get back into the superior directory.

Start

The softkey START starts reading in.
All data will be written into the previous defined directory.

Stop

The softkey STOP stops reading in.

Notes:
• While reading in it is possible to call up certain selected data by the control only from DRIVE.
• When the option "Overwrite with confirmation only" is selected, existing data will be overwritten only after confirmation.
  On reject, read-in will be continued with the next file.
• Only data with a valid extension (e.g. .MPF) can be read in (except read-in into clipboard).

A prerequisite for the data exchange is that the parameterization of transmitter and receiver is in accordance (see operating range services).

"No" tool correction values from the WinNC may be imported into the machine tool administration, since this procedure may cause system errors.

Data exchange WinNC - machine

• Set receiver to ready-to-receive state as described under "Read in data".

• Set receiver to ready-to-receive state as described under "read out data" and confirm by means of the softkey START.

• Actuate read-in on receiver by means of the softkey STOP.

EMCO

Send data

RS 232 C User	...	Drive

Select sendig target with softkey RS 232 C USER to DRIVE.

Data Out

Press the softkey DATA OUT.

You can send following data types:
• Data (tool and magazine data, R parameter, zero offsets)
• Workpieces
• Part programs
• Subprograms
• User cycles
• Standard cycles

Selection of data to be sent:

Place the cursor on the desired data type.

E.g. the cursor is on "Workpiece".
When you start transmission now, all workpieces will be sent.

Press INPUT and a list of workpieces will be displayed and you can mark a workpiece.
When you start transmission now, all data included in the workpiece (programs, subprograms etc.) will be sent.

Press INPUT again and a list of the data included in the workpiece will be displayed, etc..
The you can send single files.

With the softkey Λ you will get back into the superior directory.

Start

The softkey START starts sending data.

Stop

The softkey STOP stops sending data.

emco

Copying and pasting data from the clipboard

In the clipboard all types of data may be stored (e.g. after read-in from serial interface etc.).
You can sort these data (programs, subprograms etc.) in the corresponding directories (MPF.DIR, SPF.DIR, etc.).

Clip-
board

Press the softkey CLIPBOARD.

The cursor is placed on a file in the clipboard window (lower window).
Place the cursor on the file that should be put into the directory structure.

Place the cursor into the upper window (target window).

Place the cursor on the desired diredtory in the structure.
Into this directory the file from the clipboard will be written.

Copy and
paste

Press the softkey COPY AND PASTE.

OK

The control asks for a file name. Keep the old name or enter a new name.
Confirm with OK.

The file is copied into the target directory.

Go back to the directory overview with the softkey

∧ .

Delete

Notes:
• The files will be copied from the clipboard but not automatically deleted in the clipboard.
• Use the softkey DELETE to delete data in the clipboard.

emco

Operating Area Diagnosis

The Operating Area Diagnosis shows alarms and messages in the full length form.

With an active alarm or message change to the Operating Area Diagnosis, to get following information:

Number:
Alarm number. With several active alarms the will be displayed in time sequence.

Date:
Exact date, time when the alarm occured.

Delete criteria:
Display of the key that must be pressed to delete the alarm.

Text:
Full length text of the alarm.

Display of software versions

Press the softkey SW VERSIONS to show the versions of the individual software components.
These versions must be indicated when requesting service.

EMCO

Operating Area Start-up

The Operating Area Start-up is not active in WinNC.

For including accessories, automatisations etc. use the auxiliary software "WinConfig"

D: Programming

> **Note**
>
> This programming chapter describes all functions that can be done with WinNC.
> Depending on the machine that is operated with WinNC not all of these functions may work.
> The parameters or cycles marked with (*) are only available for cycle version 6.02.
>
> Example:
> The lathe Concept TURN 55 has no position controlled main spindle, therefore no spindle position can be programmed.

> When programming in editor, set a blank between every single word (z.B.: G0 X20 Z-35).

EMCO

Surveys

G- commands

COMMAND	MEANING
G0	Rapid traverse
G1	Machining traverse
G2	Circular interpolation clockwise
G3	Circular interpolation counterclockwise
CIP	Circular interpolation via intermediate point
G4	Dwell time
G9	Exact stop non-modal
G17	Working plane selection XY
G18	Working plane selection XZ
G19	Working plane selection YZ
G25	Minimum programmable working area limitation / programmable spindle speed limitation
G26	Maximum programmable working area limitation / programmable spindle speed limitation
G33	Thread pitch constant
G331	Tread tapping
G332	Tread tapping / retraction
G40	Cutter radius compensation OFF
G41	Cutter radius compensation ON Left
G42	Cutter radius compensation On Right
G53	Deselection settable zero offset
G54-G57	Settable zero offset
G500	Deselection zero offset
G505-G599	Settable zero offset
G60	Exact stop modal
G601	Step enable if positioning window fine reached
G602	Step enable if positioning window coarse reached
G603	Step enable if setpoint reached
G63	Tread tapping without synchronization
G64	Contouring mode
G641	Contouring mode with programmable transition rounding
G70	Input system inch
G71	Input system metric
G90	Absolute dimensioning
G91	Incremental dimensioning
G94	Feed in mm/min, inch/min
G95	Rotational feed rate in mm/rev, inch/rev
G96	Constant cutting speed ON
G97	Constant cutting speed OFF
G110	Pole parameter, with reference the last approched position
G111	Pole parameter, absolute in the workpiece coordinate system
G112	Pole parameter, with reference to the last valid pole
G140	Soft approach and leaving
G141	Approch from the left and/or leaving from the left

EMCO

COMMAND	MEANING
G142	Approach from the right and/or leaving from the right
G143	Direction of approach and/or leaving according to the relative position from start and/or end point to the tangent direction
G147	Approach with a straight line
G148	Leaving with a straight line
G247	Approach with a quarter circle
G248	Leaving with a quarter circle
G340	Approach and leaving in space (starting position value)
G341	Approach and leaving in the plane
G347	Approach with a semicircle
G348	Leaving with a semicircle
G450	Approach and leave contour
G451	Approach and leave contour

EMCO

M- Commands

COMMAND	MEANING
M0	Programmed stop
M1	Optional stop
M2	End of program
M2=3	Driven tools on clockwise
M2=4	Driven tools on counterclockwise
M2=5	Driven tools off
M3	Spindle on clockwise
M4	Spindle on counterclockwise
M5	Spindle stop
M6	Tool change
M8	Coolant on
M9	Coolant off
M10	Screw brake on
M11	Screw brake off
M17	Subprogram end
M20	Tailstock backward
M21	Tailstock forward
M23	Collecting tray Backward
M24	Collecting tray Forward
M25	Clamping device open
M26	Clamping device close
M30	Program end
M32	Program end for loading operation
M57	Spindle oscillate ON
M58	Spindle oscillate OFF
M67	Bar feed / loading magazine feed ON
M68	Bar feed / loading magazine feed OFF
M69	Bar change
M71	Puff blowing on
M72	Puff blowing off

EMCO

Cycles

DRILLING CYCLES	
Cycle 81	Drilling, Centering
Cycle 82	Drilling, Counterborin
Cycle 83	Deep hole drilling
Cycle 83E	Deep hole drilling
Cycle 84	Rigid tapping
Cycle 84E	Rigid tapping
Cycle 840	Tapping with compensation chuck
Cycle 85	Boring 1
Cycle 86	Boring 2
Cycle 87	Boring 3
Cycle 88	Boring 4
Cycle 89	Boring 5

TURNING CYCLES	
Cycle 93	Grooving cycle
Cycle 94	Undercut cycle
Cycle 95	Stock removal cycle
Cycle 96	Thread undercut
Cycle 97	Thread cutting cycle
Cycle 98	Chaining of threads

EMCO

Command shortcuts

COMMAND	MEANING
AC	Enter single axis position absolute
ACN	Approach circular axis position in negative direction
ACP	Approach circular axis position in positive direction
AND	Logical AND connection
AP	Polar angle
AR	Circle segment angle
AXIS	Variable type
AX	Axis operator
AXNAME	String operation
AMIRROR	Mirror coordinate system, additive
AROT	Rotate coordinate system, additive
ASCALE	Alter scale, additive
ATRANS	Programmable zero offset
B_AND B_NOT B_OR B_XOR	Bit operators
BOOL	Variable type
CASE	Vocabulary word for jump instruction
CIP	Circular interpolation with intermediate point (circle through points)
CHAR	Variable type
CHF	Insert chamfer (chamfer)
CR	Circle radius
CFC	Constant feed at contour
CFIN	Constant feed at the contour for internal radius, constant feed at tool centre for external radius (constant feed at internal radius)
CFTCP	Constant feed in tool centre point
CONTPRON	Activate contour preparation in tabular form
CHR	Insert chamfer
D	Tool offset number
DC	Approach circular axis position on shortest way (direct to contour)
DIAMOF	Radius as dimension
DIAMON	Diameter as dimension
DEF	Define variable
DISPLOF	Suppress current block display
DISPLON	Display in the program window on
DIV	Integer division
DEFAULT	Check structures
DEFINE AS	Macro programming
DISC	Compensation at outside corners Flexible proogramming of the approach and retraction instruction
DISCL	Distance of the end point of the machining level during soft approach and leaving
DISR	Distance of the milling cutter edge from the starting point during soft approach and leaving
ELSE	Check structures
ENDFOR	Check structures
ENDLOOP	Check structures
ENDWHILE	Check structures
ENDIF	Check structures
EXECTAB	Block by block execution of contour elements of a table
EXECUTE	End definition
F	Feed

EMCO

COMMAND	MEANING
FOR	Check structures
FRAME	Variable type
FAD	Speed of the slow feed movement with soft approach and leaving
GOTOB	Jump instruction with jump destination backwards
GOTOF	Jump instruction with jump destination forwards
I1	Address for circle interdot
IC	Enter single axis position incremental (incremental coordinates)
IF	Vocabulary word for condition
INT	Integers with leading sign
INTERSEC	Calculate contour preparation in tabular form
ISAXIS	Is a certain axis available (Enquiry via axis number)
J1	Address for circle interdot
KONT	Cutter radius compensation, surround contour in start and end point
K1	Address for circle interdot
LIMS	speed limitation
LOOP	Check structures
MCALL	Modal cycle or subprogram call (the following call stays active until it is deselected with MCALL without parameter) (modal call)
MSG	Message output in an alarm
MIRROR	Mirroring
N	Address of block number
NOT	Negation
NORM	Cutter radius compensation, direct contour approach (normal approach)
OFFN	Offset countur-normal
OR	Logical OR connection
P	Program repeat
PM	Indication of the feed FAD for approaching and departing as linear feed FAD=PM(...)
PR	Indication of the feed FAD for approaching and departing as feed per revolution
PROC	Subprogram start
R	R-parameter R[0]-R[99]
ROT	Rotate coordiante systen (rotation)
REAL	Real number
RET	Subprogramm retreat
RND	Insert radius at contour corner (round)
RNDM	Insert radius at contour corner modal (round modal)
RP	Polar radius (radius polar)
RPL	Rotation of plane
REP	Field initialization
S	Spindle address
SAVE	Saving the register at subprogramm calling
SETAL	Alarms are programmed by inserting this keyword
SET	Initialization of value lists
SETMS	Define master spindle
SF	Starting point offset, only needed for multiple threads
SPCOF	Position controlled spindle operation off
SPCON	Position controlled spindle operation on
STRING	Variable type
SCALE	Alter scale factor (scale)
STRLEN	String operation
SPOS	Spindle position
SPOSA	Spindle position
SUBSTR	Define the part of a string

EMCO

COMMAND	MEANING
SUPA	Blockwise suppression of all zero offsets (suppress all)
SBLOF	Single set suppression on
SBLON	Single set suppression off
TRANS	Programmed zero offset (translation)
T	Tool address
TURN	Number of turns for helical line (turns)
TRAFOOF	Deaktivates an active transformation (transformation off)
TRACYL	XZ- plane transformation
TRANSMIT	XY-plane transformation
UNTIL	Check structures
VAR	Define variable
WAITP	Wait for the axis to finish traversing
WAITS	Wait for spindle position to be reached
WHILE	Check structures
WALIMOF	Working field limiting off
WALIMON	Working field limiting on
XOR	Exklusiv OR

Arithmetic functions

COMMAND	MEANING
+, -, *, /, %, ^	Arithmetic function
SIN()	Sine function
COS()	Cosine function
TAN()	Tangent function
ASIN()	Arc sine function
ACOS()	Arc cosine function
ATAN()	Arc tangent function
ATAN2(,)	Arc tangent function (section X, section Y)
SQRT()	Root function
POT()	Power function
SQR()	Squaring function
EXP()	Exponential function (base e)
LOG()	Logarithm function (base e)
LN()	Natural logarithm function
PI	Circle partition number (3.141592...)
TRUE	logic True (1)
FALSE	logic False (0)
ABS()	Absolute function
TRUNC()	Integer number part function
ROUND()	Rounding function
MOD()	Modular function

EMCO

Calculator

Mathematical expressions can be directly evaluated in an input field.
You can use any number of parentheses in the mathematical expressions.
To calculate the terms, please press the "Enter" key or leave the input field.

In case errors occur during the formula evaluation, the term having been entered last will be displayed and WinNC will release an error message.

The currently displayed value can be used in a formula with $\boxed{\begin{smallmatrix}P\\=\end{smallmatrix}}$.

The formula being entered last can be edited again by means of the key "?".

COMMAND	DESCRIPTION	EXAMPLE	RESULT
+	addition	1+1	2
-	subtraction	3-2	1
*	multiplication	5*3	15
/	division	15/3	5
%	modulus (remainder)	10%4	2
^	power	5^2	25
PI	circular graduation number	PI	3,141593
SIN( )	sine	SIN(90)	1
ASIN( )	arc sine	ASIN(-1)	-90
COS( )	cosine	COS(90)	0
ACOS( )	arc cosine	ACOS(-1)	180
TAN( )	tangent	TAN(45)	1
ATAN( )	arc tangent (value)	ATAN(1)	45
ATAN2(,)	arc tangent (X segment, Y segment)	ATAN2(0,1)	0
EXP ( )	exponential function (base e)	EXP(1)	2,718282
LOG ( )	logarithm (base e)	LOG(5)	1,609
SQRT ( )	square root	SQRT(2)	1,414
MOD (,)	modulus function	MOD(10,4)	2
TRUE	true	TRUE	1
FALSE	false	FALSE	0
AND	conjunction	1AND1	1
OR	disjunction	1OR1	1
NOT	negation	NOT(1OR1)	0

Functions of the calculator

emco

System variable

System variable	Description
$P_AXN1	Current address of the geometry axis- abscissa
$P_AXN2	Current address of the geometry axis- ordinate
$P_AXN3	Current address of the geometry axis- applicate
$P_IFRAME	Current settable frame
$P_PFRAME	Current programmable frame
$P_BFRAME	Actual base frame variable
$P_ACTFRAME	Current total frame
$P_UIFR[]	Adjustable frames (G54)
$P_F	Path feed F last programmed
$P_DRYRUN	0 (FALSE): Dry run on 1 (TRUE): Dry run off
$P_SEARCH	1 (TRUE): Block search (with or without calculation) is active
$P_TOOLR	Active tool radius (total)
$P_TOOLNO	Active tool number T0 - T32000
$AC_MSNUM	Master spindle number
$MN_SCALING_SYSTEM_IS_METRIC	Ground system Metric (1: Metric, 2: Inch)
$MN_SCALING_VALUE_INCH	conversion faktor from Metric to Inch (25,4)
$MN_INT_INCR_PER_MM	Computing precision of the linear positions
$MN_MIRROR_REF_AX	Reference axis for frame elements
$P_SIM	1 (TRUE): Simulation in progress
$P_SDIR[]	Last spindle rotational direction to be programmed
$P_GG	Current G function of a G group lind. As for PLC interface
$P_EP	Setpoint last programmed
$MA_SPIND_ASSIGN_TO_MACHAX	Allocation of the spindle to the machine axis
$MA_NUM_ENCS	Number of transducers
$AA_S	Actual spindle speed: Sign corresponds to direction of rotation
$MC_DIAMETER_AX_DEF	Plane axis number
$P_AD[]	Active tool offset
$P_TOOL	Active tool edge D0 - D9
$P_MC	0 (FALSE): No modal subprogram call 1 (TRUE): Modal subprogramm call
$P_TOOLL	Avtive overall tool length
$A_IN[]	Digital input NC (1-16)
$A_OUT[]	Digital output NC
$A_INA[]	Analog input NC (1-4)

EMCO

Working Movements

> Before programming, define the origin of the workpiece coordiate system with G111.

G0, G1 Linear interpolation (cartesian)

G0: Traverse with rapid feed e.g. for fast positioning
G1: Traverse with programmed feedrate F, e.g. machining the workpiece

Format

G0   X..   Z..
G1   X..   Z..   F..

G0, G1 Linear interpolation (polar)

Format

G0   AP..   RP..
G1   AP..   RP..

Insert chamfer, radius

Chamfers or radii can be inserted between straights and arch in any combination.

Format

G..   X.. Z..   CHR=..   chamfer
G..   X.. Z..   CHF=..   chamfer
G..   X.. Z..   RND=..   radius
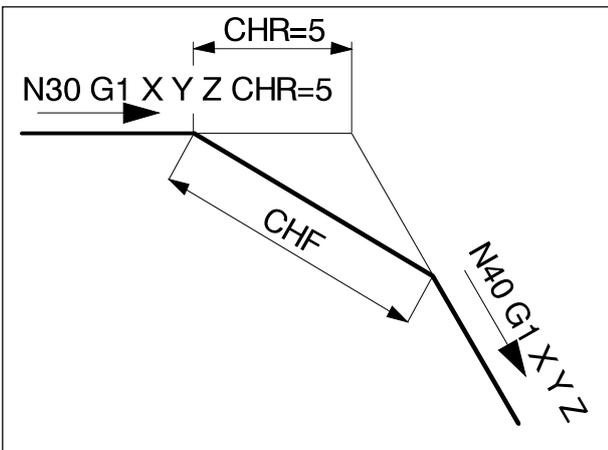
Chamfer

The chamfer will be inserted after the block in wich it was programmed. The chamfer is always in the working plane (G17).
The chamfer will be inserted symmetrically into the contour corner.
The CHR value is the length of the chamfer.

Example:

N30   G1   X..   Z..   CHR=5
N35   G1   X..   Z..
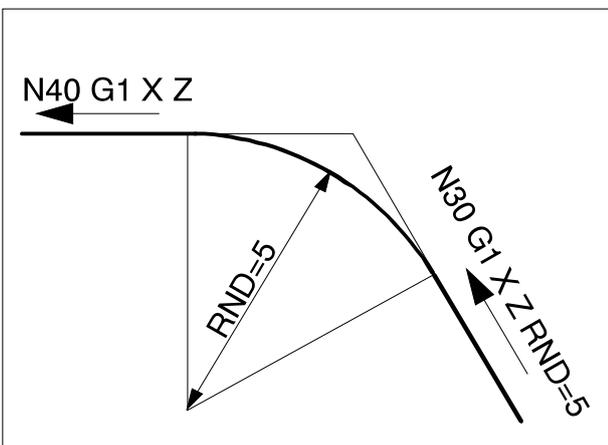
Radius

The radius will be inserted after the block in wich it was programmed. The radius is always in the working plane (G17).
The curve is an circular arc and will be inserted into the contour corner with tangentional connections.
The RND value is the radius of the arc.

Example:

N30   G1   X..   Z..   RND=5
N35   G1   X..   Z..

Modal radius RNDM

At every following contour corner a radius will be inserted until the modal radius is deselected with RNDM=0.

G2, G3, CIP Circular Interpolation

G2   clockwise
G3   counterclockwise
CIP   via intermediate point (**CI**rcle through **P**oints)
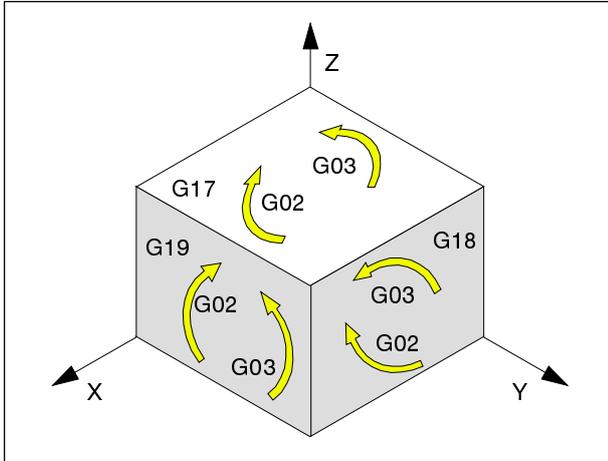
Display of the circular movement in the different planes.
For a circular movement the start and end point are in one plane (level).

Programming with start point, end point, centre point

G2/G3   X..   Z..   I..   K..

X, Z   end point E in cartesian coordinates
I, K   circle centre point M in cartesian coordinates, related to start point S
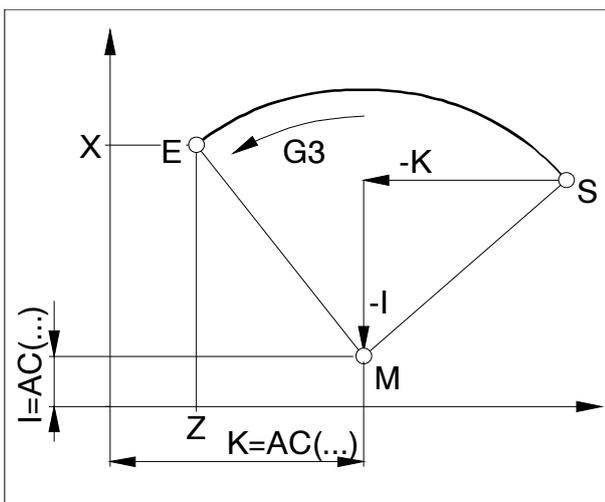
Start point
The start point is the position of the tool at the time of the call-up of G2/G3.

End point
The end point is programmed with X, Z.

Circle centre point
The circle centre point is programmed incremental with I, K based on the start point or with I=AC(..), K=AC(..) absolute from the workpiece zero point.
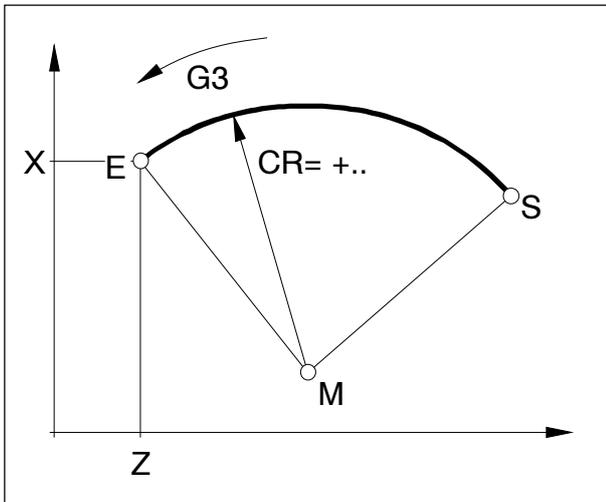
Programming with start point, end point, circle radius

G2/G3   X..   Z..   CR=±..

X, Y, Z   end point E in cartesian coordinates
CR=±   circle radius

Start point
The start point is the position of the tool at the time of the call-up of G2/G3.

End point
The end point is programmed with X, Z.

Circle radius
The circle radius is indicated with CR. The sign indicates whether the circle arc is smaller or larger than 180°.
CR=+   angle smaller or equal 180°
CR=-   angle larger 180°.
Full circles can not be programmed with CR.

Programming with start point, circle centre point or end point, spread angle

G2/G3   X..   Z..   AR=..   or
G2/G3   I..   K..   AR=..

X, Z   end point E in cartesian coordinates
         or
I, K   circle centre point M in cartesian coordinates, related to start point S
AR=   spread angle
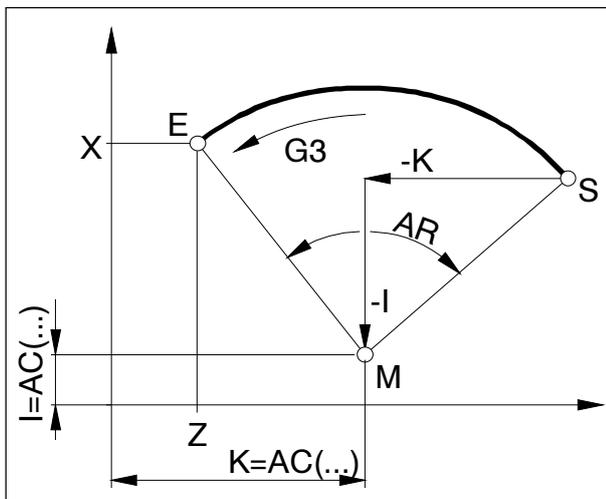
Start point
The start point is the position of the tool at the time of the call-up of G2/G3.

End point
The end point is programmed with X, Z.

Circle centre point
The circle centre point is programmed incremental with I, J, K based on the start point or with I=AC(..), K=AC(..) absolute from the workpiece zero point.

Spread angle
The spread angle must be smaller than 360°.
Full circles can not be programmed with AR.
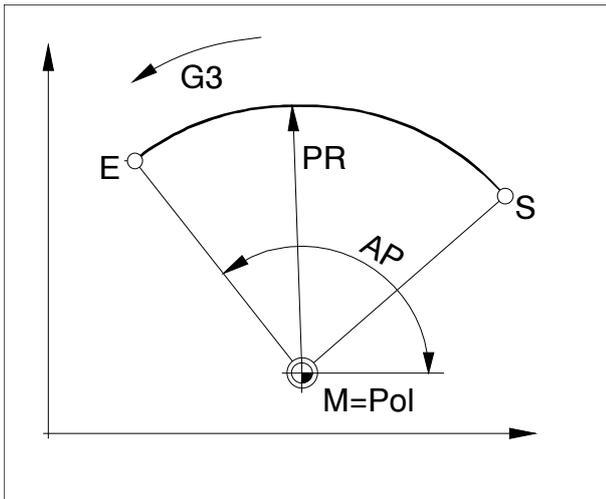
Programming with polar coordinates

G2/G3   AP=.. RP=..

AP=   end point E polar angle,
             the pole is the circle centre point
RP=   polar radius, at the same time circle radiuss

The pole of the polar coordinate system must be at the circle centre point (previous put on circle centre point with G111)
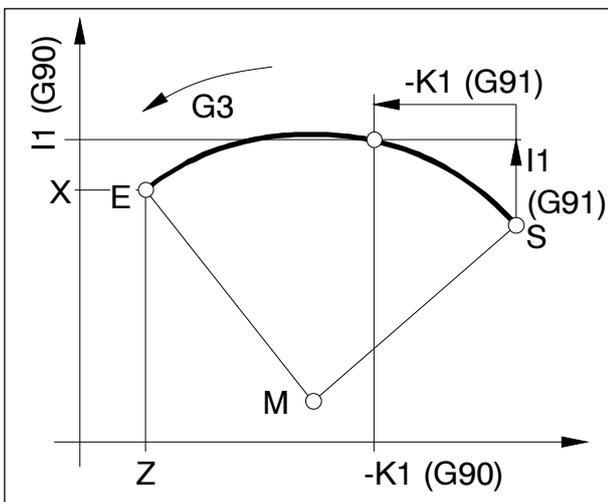
Programming with start point, intermediate point, end point

CIP   X..   Z..   I1=..   K1=..

X, Z   end point E in cartesian coordinates
I1, K1   intermediate point Z in cartesian coordinates

Start point
The start point is the position of the tool at the time of the call-up of G2/G3.

End point
The end point is programmed with X, Z.

Intermediate point
The intermediate point is programmed with I1, K1.
With G91 (incremental programming) the intermediate point is related to the start point.

G4 Dwell time

Format

N... G04 F...   [sec]
N... G04 S...   [U]

F   dwell time in seconds
S   dwell time in number of main spindle revolutions

The tool will be stopped in the last reached position
- sharp edges - transitions, clean drill / groove ground,
exact positioning.

Note
• The dwell time starts after the feed rate of the
  previous block has become 0.
• S and F are used as time values in the G4 block
  only. A previous programmed feedrate F or spindle
  speed S will be kept.

Example
N75 G04 F2.5   (dwell time = 2,5 sec)

EMCO

Exact positioning active
(G9, G60)

Exact positioning not
active (G64, G641)
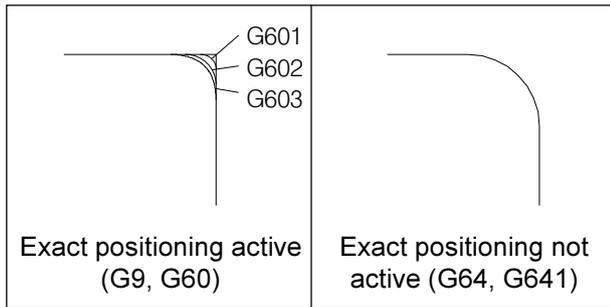
G9, G60, G601, G602, G603 Exact positioning

G9   Exact positioning, effective blockwise
G60   Exact positioning, modal
G601   Step enable if positioning window fine reached
G602   Step enable if positioning window coarse reached
G603   Step enable if end of interpolation reached

G601/G602/G603 are effective only with active G60 or G9.
The commands G64, G641 - Contouring mode deselect G60.

G9/G60:
Activation of G601, G602 or G603.
G9 is effective in the block only in which it was programmed, G60 is effective as long as it will be cancelled with G64 or G641.

G601, G602:
The following block will be started only after in the G9 / G60 the slides were stopped to standstill (short standstill delay at block end).
By that the corners will not be rounded and the transitions will become sharp.
The target position can be in a fine (G601) or coarse (G602) tolerance window.

G603:
The block change will be triggered in that moment whe the control calculates the nominal feedrate 0 for the included axes (no standstill). At this point in time the actual value is back for a run after amount. By that the edge will be rounded.
With G603 the edge will be rounded most.

G64, G641 Contouring mode

G64	Contouring mode
G641	Contouring mode with programmable rounding
ADIS=	Rounding tolerance for movements with working feed rate
ADISPOS=	Rounding tolerance for movements with rapid feed (G0)

The contour will be produced with path feed rate as constant as possible.
This results in shorter machining time and rounded transitions.

At tangential transitions the tool traverses with a path feed rate as constant as possible, in corners the feed rate will be reduced correspondingly.

Th larger the feed rate F the larger is rounding of the edges (contour error).

The command G641 allows to indicate the rounding distance.

Example:

G641   ADIS=0.5   G1   X...   Z...
Rounding must start at the earliest 0,5 mm before the corner and must be finished at the latest 0,5 mm after the corner.

G641 ADIS = 0.5

0,5

0,5

D 17

emco
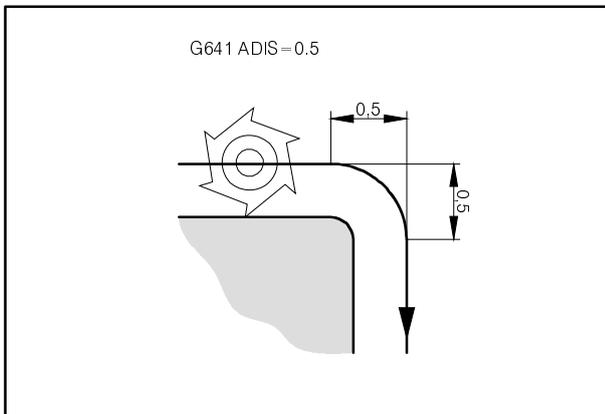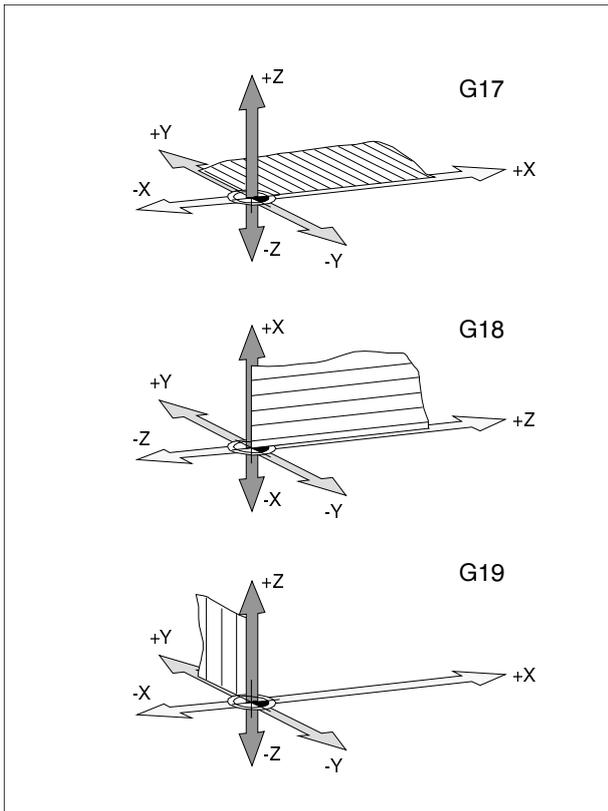
G17, G18, G19 Working plane selection

Format
N... G17/G18/G19

G17 plane XY: plane face milling (TRANSMIT), Axial drilling with original Siemenscyclen

G18 plane ZX: contour turning

G19 plane YZ: contour milling at the surface area. (TRACYL), Radial drilling original Siemenscyclen

G17-G19 selects the working plane.

- The tool axis is vertical to the working plane.
- Circle interpolation G2/G3/CIP occurs in the working plane.
- Polar coordinate interpolation occurs in the working plane.
- Cutter radius compensation G41/G42 occurs in the working plane.
- The infeed movements e.g. for drilling cycles are vertical to the working plane.

EMCO
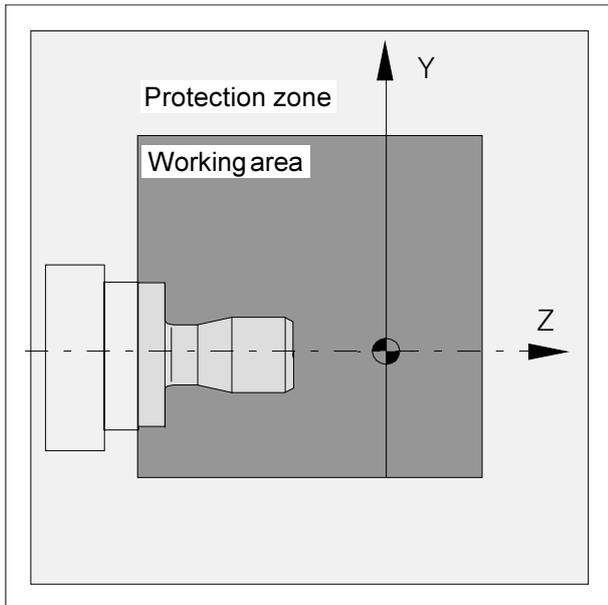
G25, G26 Programmable working area limitation

Format
N... G25/G26   X...   Z...

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.

G25 and G26 must be programmed in a separate program block.
The programmable working area limitation will be defined in the program by G25 and G26, and switched on and off with WALIMON and WALIMOF.

G25	Lower working area limitation
G26	Upper working area limitation
WALIMON	Working area limitation on
WALIMOF	Working area limitation off

G25, G26 Programmable spindle speed

Format
N... G25/G26   S...

A minimum and a maximum spindle spee can be defined with G25/G26 .

G25 and G26 must be programmed in a separate program block.
The programmable spindle speed limitation with G25/G26 overwrites the values in the setting data and therefore is kept also after program end.

G25	Lower spindle speed limitation
G26	Upper spindle speed limitation
S	Min / max spindle speed

EMCO
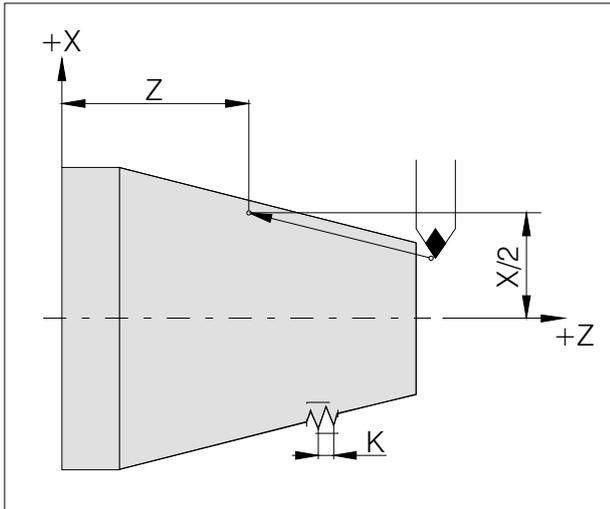
G33 Thread cutting

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

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

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.
Chaining of threads is carried out by programming G33 blocks in direct succession (without traversing movements between the threads).
Prior to the first G33 block, the number of threads has to be programmed with SETTHREADCOUNT (n).

Note
• Feed and spindle override are not active with G33 (100%).
• A fair sized undercut has to be machined first.

Example chaining of threads:
N010 SETTHREADCOUNT (3)
N011 G33 X... Z... I/K... SF...
N012 G33 X... Z... I/K... SF...
N013 G33 X... Z... I/K... SF...
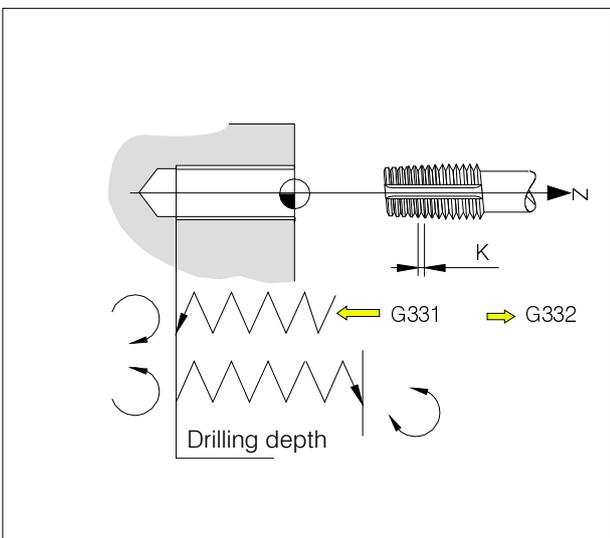
G331/G332 Tapping without compensation chuck

Format
N... G331 X... Z...   K...
N... G332 X... Z...   K...

X, Z .... Drilling depth (End point)
K .... Thread lead

Drilling depth, Thread lead
Drilling in Z- direction, Thread lead K

G331 Tapping:
Tapping is described by the drilling depth (end point of the thread) and the lead.

G332 Retraction movement:
This movement is described with the same lead as the G331 movement. The reversal in the direction of the spindle is performed automatically.

SPOS have to be used to prepare the spindle for tapping.

EMCO

G63 Thread tapping with compenstion chuck

Format

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

Thread tapping without synchronisation.

Programmed spindle speed S, programmed feed rate F and pitch P of the tap must match:
F [mm/min] = S [U/min] x P [mm/U] resp.
F [mm/U] = P [mm/U]

The dive-in movement of the tap is programmed with G63.
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

Cutter Radius Compensation G40-G42

G40   Cutter radius compensation OFF
G41   Cutter radius compensation LEFT
G42   Cutter radius compensation RIGHT
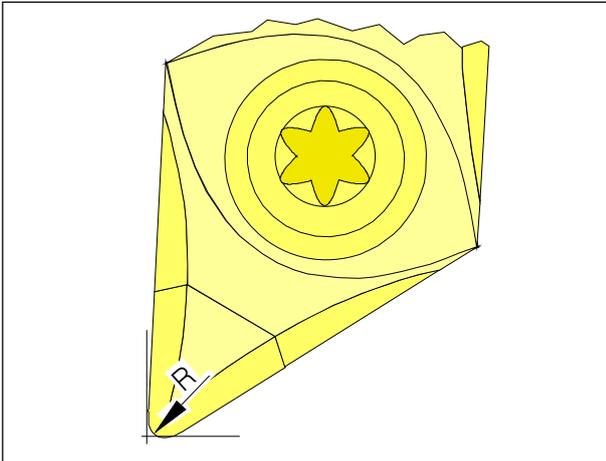
Tip radius and theoretical cutting point

With tool measuring the tool tip is measured only at two points (tangential to X and Z axis).
Therefore the tool correction describes only a theoretical cutting point.
This point is traversed along the programmed path.

While traversing along the axis direction (longitudinal or face turning) the tangential points at the tool tip are cutting.
There are no dimensional errors at the workpiece.

With simultanuous traversing in both axes (taper, radius) the position of the theoretical cutting point does not match the position of the real cutting point.
Dimensional errors on the workpiece will occur.
Maximum contour error with machining 45° movements without cutter radius compensation:
Cutter radius 0,4 mm 0,16 mm path error 0,24 mm error in X and Z.

With using the cutter radius compensation the control automatically compensates these errors.

Cutting movements parallel and oblique to the axes

G40   Cutter radius compensation OFF

Cutter radius compensation will be deselected by G40.
Deselection is allowed only in combination with a straight movement (G00, G01) (Departing movement).
G40 can be programmed in the same block as G00 or G01 or in the previous block.
G40 is normally programmed with retraction to the tool change point.
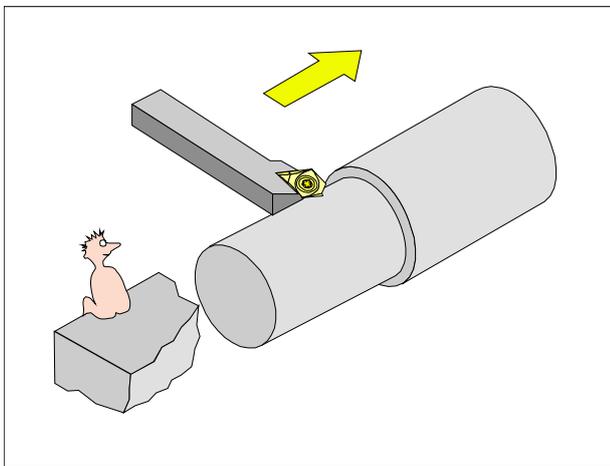
Definition G41 Cutter radius compensation left

G41   Cutter radius compensation left

When the tool is **left** of the contour to be machined (viewed in direction of feed), G41 must be programmed.

Notes
- Direct change between G41 - G42 is not allowed - previous deselection with G40.
- Predeterminating the tool radius R and cutter position (tool type) in the tool data register is necessary.
- Selection must occur in combination with G0 or G1 (Approaching movement).
- No change of tool correction while active cutter radius compensation.
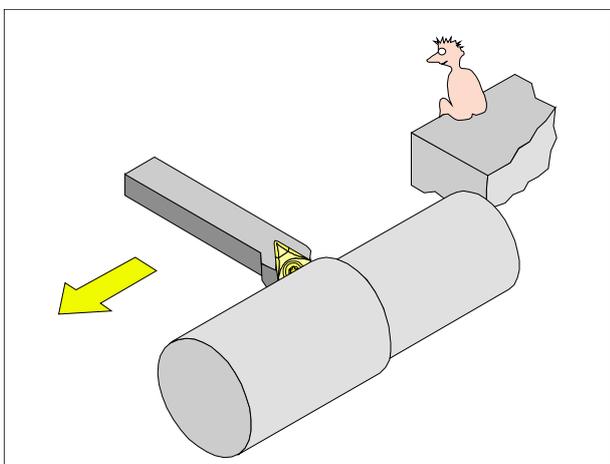
Definition G42 Cutter radius compensation right

G42   Cutter radius compensation right

When the tool is **right** of the contour to be machined (viewed in direction of feed), G42 must be programmed.

Notes see G41!
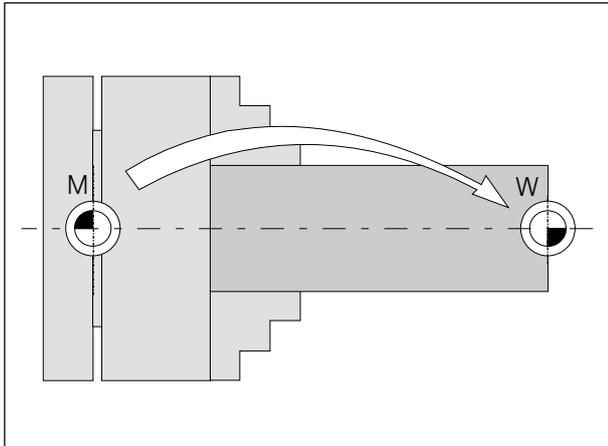
Zero offsets G53-G57, G500-G599, SUPA

G53	The zero offsets will be suppressed for one block.
G500	G54 - G599 are switched off.
G54-57	Settable zero offsets.
G505-599	Settable zero offsets.
SUPA	Blockwise suppressing, including programmed zero offsets and DRF offsets (**SUP**press **A**ll)

Zero points are used to indicate the position of the workpiece on the machine.

Normally the coordinate system is shifted with G54-G599 to a stop point (W_1) at the clamping device (fix stored), and the following shift to the workpiece zero point (W_2) occurs with TRANS (variable).

Inch dimensions G70, Metric dimensions G71

Depending on G70 / G71 the following values can be entered in inch or mm:
- Path informationen X, Y, Z
- Circle parameter I1, K1, I, K, CR
- Thread pitch
- Programmable zero offsets TRANS, ATRANS
- Polar radius RP

All other values e.g. feed rates, tool corrections or settable zero offsets will be calculatet in that unit that was prese in the machine data.

EMCO

Coordinaten, Zero Offset

Working plane G17-G19

In the working plane the tool radius is effective, vertical to the working plane the tool length.

Main working plane for turning: G18 (ZX)

The entire zero point offset effective in a workpiece program is the sum of basis zero point offset + adjustable zero point offsets + frames.
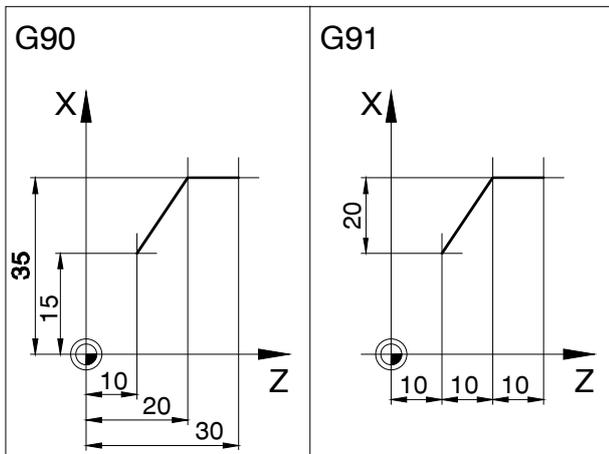
G90 Absolute dimensions

The dimensions refer to the actual zero point.
The tool traverses **ON** a programmed position.

G91 Incremental dimensions

The dimensions refer to the last programmed tool position.
The tool traverses **FOR** a distance to the next position.

Single axes can be programmed absolute or incremental without respect to G90 / G91.

Examples:
G90
G0 X40 Z=IC(20)
Here is the Z value incremental although G90 Absolute dimensions is active.

G91
G0 X20 Z=AC(10)
Here is the Z value absolute although G91 Incremental dimensions is active.

Feed Programming G94, G95

General

- Feed values are not influenced by G70/71 (inch-mm), only the machine setting data is valid.
- After every change G93-95 F must be programmed again.
- The feed rate F is valid for path axes only, but not for synchronous axes.

Feed rate in mm/min G94

Slide movement X, Z:
The address F is the feed rate in mm/min.
Main application for milling.

Vorschub in mm/U G95

Slide movement X, Z:
The address F is the feed rate in mm/rev of the milling spindle.
Main application for turning.

If G95 is not programmed a feed rate F must be programmed.

Constant cutting speed G96, G97, LIMS

G96   Constant cutting speed ON
S   Cutting rate m/min
G97   Constant cutting speed OFF
LIMS pindle speed limitation with active G96

When G96 is active, the spindle speed, depending on the respective workpiece diameter, is autoomatically modified in order that the cutting rate S in m/min remains constant tool edge.
Thus increases the uniformity and thus the surface quality of turned parts.

If you machining a workpiece that varies greatly in diameter, it is advisable to specifiy a speed limit for the spindle. This prevents excessively high speeds with small diameters.
LIMS active with G96

Example
N10 G96 S100 LIMS=2500

Polar coordinates G110-G112

With polar coordinate programming the positions will be indicated as angle and radius, related to the pole (zero point of the polar coordinate system).
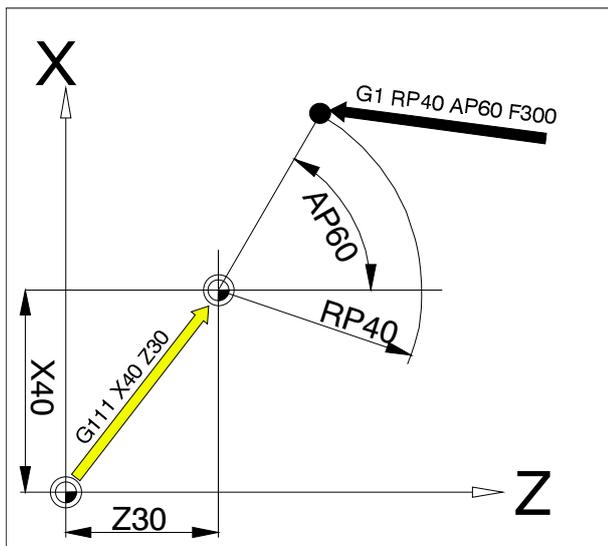
Determination of the pole

G110 Pole position related to the last programmed tool position.

G111 Pole position, related to the actual zero point of the workpiece coordinate system.

G112 Pole position related to the last valid pole.

The pole can be indicated in cartesian or polar coordinates.

X,Z   coordinates of the pole (cartesian)
RP   polar radius (= distance pole - target)
AP   polar angle between distance pole - target and angle reference axis (first programmed pole axis)

Example

G111   Z30   X40
G1   RP=40 AP=60 F300

G111 puts the pole on the absolute position 30/40.

G1 moves the tool from its previous position to the polar coordinate RP40/AP60.
The angle relates to the Z axis, because Z was programmed first in the G111 block.

EMCO

G147

G247

G247

Approach and leaving movement described with interdot P_3 (with simultaneous activation of the tool radius correction).
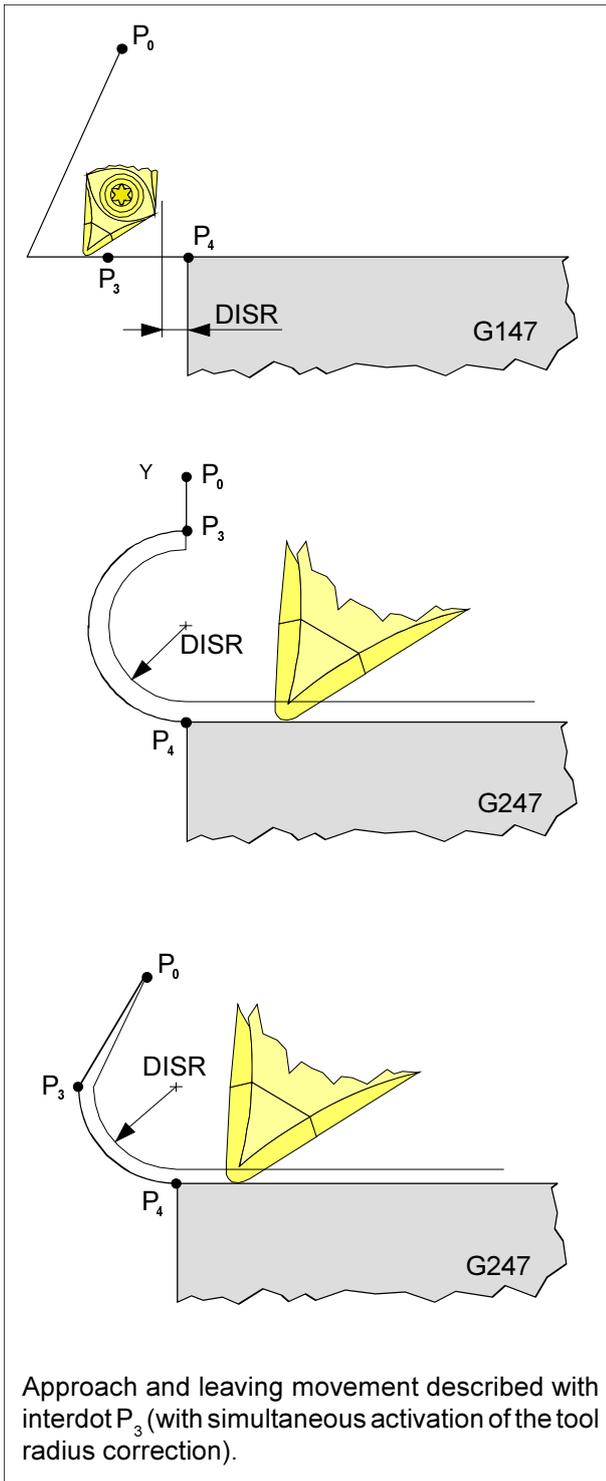
Traversing movements with G0/G1 have to be programmed prior to the soft departure and approach.
The programming of G0/G1 is not possible in the same block.

Soft approach and leaving G140 - G341, DISR, DISCL, FAD

G140 Soft approach and leaving
G141 Approach from the left and/or leaving from the left
G142 Approach from the right and/or leaving from the right
G143 Direction of approach and/or leaving dependent
    on the relative position of start and/or end point
    to the tangent direction
G147 Approach with a straight line
G148 Leaving with a straight line
G247 Approach with a quarter circle
G248 Leaving with a quarter circle
G340 Approach and leaving in space (starting position value)
G341 Approach and leaving in the plane
G347 Approach with a semicircle
G348 Leaving with a semicircle
G450 Approach and leave contour
G451 Approach and leave contour
DISR • Approach and leaving with straight line, distance of the milling cutter edge from starting point to contour
    • Approach and leaving with circles. Radius of the tool centre path
DISCL Distance of the end point of the rapid motion of the machining level

DISCL=AC Indication of the absolute position of the end point of the rapid movement
DISCL=0   G340: P_1, P_2, P_3 coincide
          G341: P_2, P_3 coincide
FAD   Indication of the feed speed
          G341: from P_2 to P_3
          G340: from P_2 and/or P_3 to P_4
FAD=PM   Linear feed (like G94)
FAD=PR   Rotation feed (like G95)

The function soft approach and leaving serves to approach the starting point of a contour tangentially irrespective of the position of the initial point.
The function is mainly used in connection with the tool radius correction, however, it is not obligatory.

The approach and leaving movement consists of a max. of 4 partial movements:
• starting point of the movement (P_0)
• interdots (P_1,P_2,P_3)
• end point (P_4)

The points P_0, P_3, and P_4 are always defined. The interdots P_1 und P_2 can be omitted according to the machining conditions.

EMCO

feed movement

P₀   P₁

straight line,
circle or helix

P₂ P₃

DISCL

P₄

G340

machining level

straight line
or circle

P₀   P₁

P₂

DISCL

P₃   P₄

G341

Start movement dependent on G340/G341
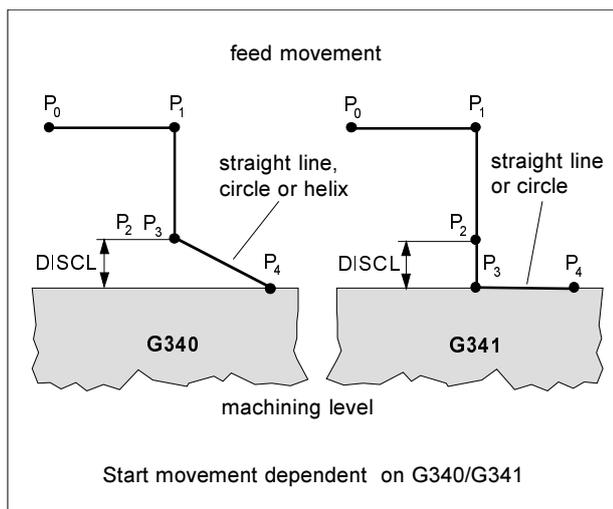
Selection of the approach and/or leaving direction
Determination of the approach and leaving direction
by means of the tool radius correction

with positive tool radius:
G41 active - approach from the left
G42 active - approach from the right

Division of the movement from the starting to the end point (G340 and G341)
The characteristic approach of P_0 to P_4 is represented
in the adjoining figure.

In those cases, in which the position of the active
levels is programmed with G17 to G19, a possibly
active rotating FRAME is taken into consideration.

EMCO

Approach Characteristic NORM, KONT

Approach and leave contour NORM/KONT

NORM: The tool approaches straight and stands vertcal to the start point.
If the start / end point is not on the same side of the contour as the first / last point of the contour a contour violation will happen.

KONT: The tool traverses around the contour point as programmed with G450/451.
G450: surround with arc
G451: surround with straight

— — — programmed tool path
——————— real tool path with correction
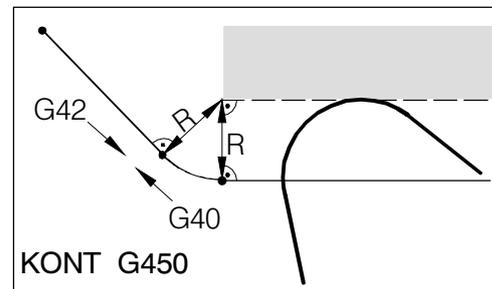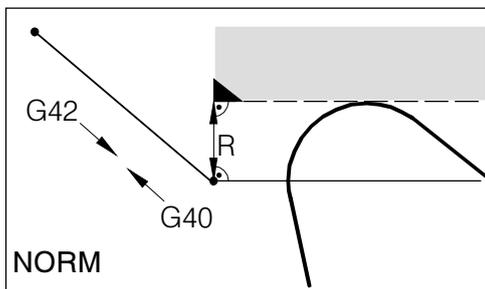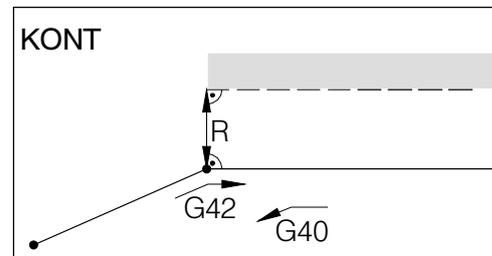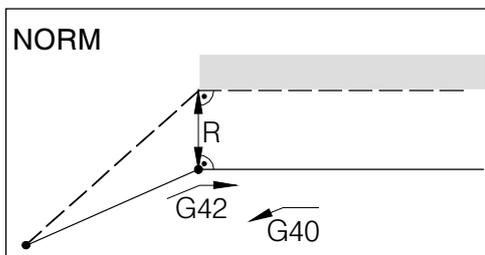
With approaching or leaving with NORM a contour violation (black) will happen, if the start or end point is behind the contour.

With approaching or leaving with KONT the tool traverses around the corner with an arc (G450) or a straight (G451).

EMCO

Cycle call

Cycle call occurs as following:

Cycle (Parameter 1, Parameter 2, ...)

In the survey pictures and in the cycle description you will find all needed parameter for the cycles.

In the call-up of the cycles the only parameter values will be enterd (no parameter term).
Therefore the sequence of parameter must be kept, for that the values can not be misinterpreted.
When a parameter is not needed, on its place a additional comma must be given.

Example:

A drill hole should be made with CYCLE 81. A safety distance is not necessary (e.g. there is already a groove in the workpiece).
The hole should be 15 mm deep, related to the zero point.

CYCLE81 (5,0,,-15)

CYCLE 81 . Drilling, centering
5 .... Retraction plane 5 mm over zero point (= workpiece surface)
0 .... Reference plane at zero level
,, ... Here the safety distance would be programmed. For that the control knows that the next value is the drilling depth, an additional comma is set.
-15 .... End depth absolute
) .... The parameter DPR is not indicated. Since no further parameter follows, no additional comma is necessary.

Notes for cycle description

The cycle description starts with a survey table containing the cycles and their parameter.
Following is the exact description of the parameter.

In the survey table the cycles are based on the previous cycle, that means only those parameter are described that are different to the previous cycle or new.

Example:

CYCLE 82 has the same parameter as CYCLE 81, only parameter 6 DTB is added.
CYCLE 83 has the first 5 parameter like CYCLE 81, the parameter 6 - 12 are added.
CYCLE 84 has the parameter 1-5 like CYCLE 81, the parameter 6 like CYCLE 82 and the parameter 7 - 12 are added, etc..

You can also call cycles with MCALL. (see "Modal subprogram MCALL")

Drilling Cycles

Cycle 81	Drilling, Centering
Cycle 82	Drilling, Counterborin
Cycle 83	Deep hole drilling
Cycle 83E	Deep hole drilling
Cycle 84	Rigid tapping
Cycle 84E	Rigid tapping
Cycle 840	Tapping with compensation chuck
Cycle 85	Boring 1
Cycle 86	Boring 2
Cycle 87	Boring 3
Cycle 88	Boring 4
Cycle 89	Boring 5

CYCLE81 Drilling, Centering
CYCLE82 Drilling, Counterboring

CYCLE81 (RTP,RFP,SDIS,DP,DPR)
CYCLE82 (RTP,RFP,SDIS,DP,DPR,DTB)

RTP   retraction plane absolute   **ReT**raction **P**lane
After the cycle the tool is placed on this height. RTP must be higher than the reference plane.

RFP   reference plane absolute   **ReF**erence **P**lane
Height of the workpiece surface, most time the workpiece reference point is on the surface (RFP=0)

SDIS   safety distance without sign   **S**afety **DIS**tance
The tool traverses with rapid feed until SDIS over reference plane and then changes to working feed

DP   end depth absolute   **DeP**th
Depth of the hole, related to the workpiece reference point.

DPR   end depth relative to reference plane   **DeP**th **R**elative
Depth of the hole, related to the reference plane, without sign.

Either DP or DPR can be programmed. If nevertheless both parameters are programmed, DPR will be valid.

DTB   dwell time at hole ground in [s]   **D**well **T**ime at end drilling de**P**th
The tool retracts only after a dwell time to clean the hole ground (only CYCLE82).
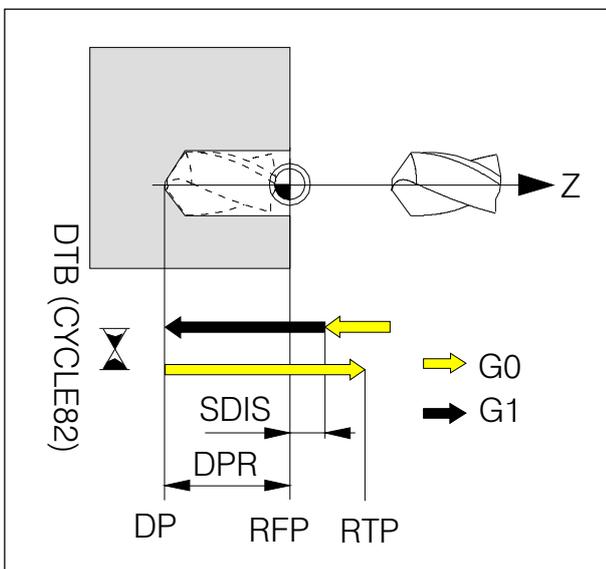
Previous to the cycle the tool must be placed over the hole position (X=0).

The tool drills with programmed feed to hole depth DP/DPR and retracts with rapid feed.

CYCLE82 allows a dwell time DTB at hole ground.

Drilling on the main spindle

Return plane, absolute 5
Reference plane absolute 0
Safety distance 2
Final drilling absolute .... -20
Depth inc. 0
Dwell time (only Cycle 82) 0
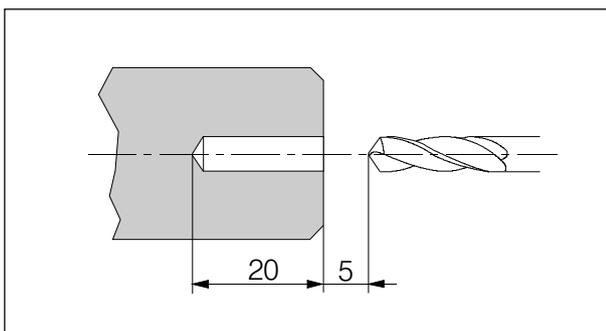
```
G54
TRANS Z70
G17
T8 D1
G95 S1000 M3 F0.12
G0 X0 Z5
Cycle 81 (5, 0, 2, -20, 0)
G0 X100 Z10
G18
M30
```


D 34

Drilling with driven tools (axial)

Return plane, absolute 5
Reference plane absolute 0
Safety distance 2
Final drilling absolute .... -20
Depth incr.   0
Dwell time (only Cycle 82) 0

Drilling with driven tools (radial)

Return plane, absolute 32
Reference plane absolute 30
Safety distance 2
Final drilling absolute .... -5
Depth incr.   0
Dwell time (only Cycle 82) 0
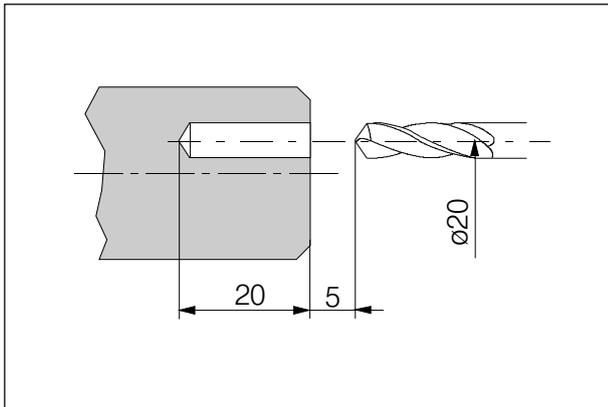

```
G54
TRANS Z70
G17
T7 D1
SPOS[1] =0
SETMS(2)
G95 S1000 M3 F0.12
G0 X20 Z5
Cycle 81 (5, 0, 2, -20, 0)
G0 X100 Z20
M5
SETMS(1)
G18
M30
```
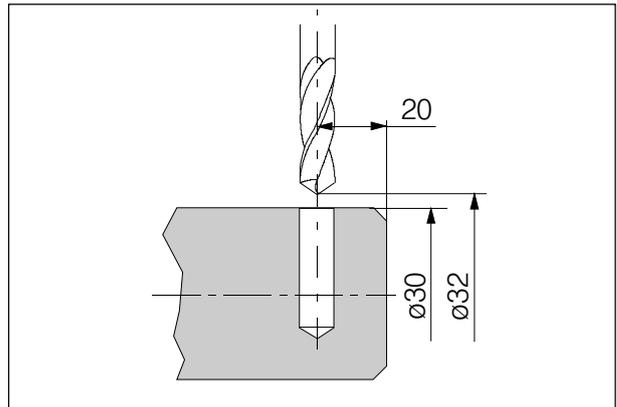
```
G54
TRANS Z70
G19
T5 D1
SPOS[1] =0
SETMS(2)
G95 S1000 M3 F0.12
G0 X32 Z-20
Cycle 81 (32, 30, 2, -5, 0)
G0 X50 Z20
M5
SETMS(1)
G18
M30
```

EMCO

CYCLE83 Deep hole drilling

CYCLE83 (RTP,RFP,SDIS,DP,DPR,FDEP,FDPR,DAM,
        DTB,DTS,FRF,VARI,AXN,MDEP,VRT,DTD,DIS1)

Parameter additional to Cycle82:

FDEP	1st drilling depth absolute	**F**irst **DEP**th
FDPR	1st drilling depth relative	**F**irst **D**e**P**th **R**elative
DAM	degression amount	**D**egression **AM**ount
DTS	dwell time before infeed in [s]	**D**well **T**ime at infeed **S**tart
FRF	feed reduction factor for 1st infeed	**F**eed **R**eduction **F**actor
VARI	machining variant	**VARI**ant
AXN(*)	Tool axis	**AX**is
MDEP(*)	Minimum drilling depth	**M**in. drilling **DEP**th
VRT(*)	Variable return path	**V**ariable **ReT**urn path
DTD(*)	Dwell time at final drilling depth	**D**well **T**ime
DIS1(*)	Lead distance	**DIS**tance

Function:
The tool drills up to the requested final drilling depth at the programmed spindle speed and feed rate. The deep hole is machined by repeated incremental plunging depths - whose maximum value can be determined - until the total hole depth is reached. After every plunging depth the drill can be either retracted to the reference plane+safety distance for chip removal or it can be retracted by 1mm each time for chip breaking.

FDEP
Depth of the first drilling infeed related to the workpiece zero point.

FDPR
Depth of the first drilling infeed related to the reference plane, without sign.

DAM
Starting from the first drilling depth each of the following infeeds will be reduced each by the value DAM.

DTB
The dwell time at the final drilling depth (chip breaking) is programmed in seconds or revolutions of the main spindle.
DTB < 0 Entry in revolutions
DTB = 0 Entry in seconds

DTS
The tool is retracted after each infeed and will go forward again after the dwell tim DTS.

EMCO

FRF
With this factor FRF the programmed feed can be reduced for the first infeed, possible input: 0,001 - 1.

VARI
VARI=0 - chip breaking
After each infeed the tool retracts for 1 mm to break the chips.
VARI=1 - chip removal
After each infeed the tool retracts out of the boring to the reference plane to remoce the chips out of the boring.

AXN
Selection of the tool axis:
X=2; Z=1

MDEP
For drilling stroke calculations above a decrease factor, a minimum drilling depth can be determined. In case the calculated drilling stroke is below the minimum drilling depth, the remaining drilling depth is machined in strokes from the size of the minimum drilling depth.

VRT
Return path during chip breaking. With VRT=0 (parameter not programmed) the drill is retracted by 1 mm each.

DTD
The dwell time at the final drilling depth can be entered in seconds or in revolutions.
DTD > 0 Entry in seconds
DTD < 0 Entry in revolutions
DTD = 0 Dwell time as programmed under DTB
DIS1
The lead distance after penetrating again into the bore can be programmed (for VARI=1).
DIS1 > 0 Positioning at the programmed value
DIS1 = 0 Automatic calculation

Previous to the cycle the tool must be placed over the hole position (X=0).

The tool drills with programmed feed to the first drilling depth FDEP/FDPR, retracts in rapid, next infeed etc.. The infeed depth will be reduced each for DAM.

Drilling on the main spindle

Return plane, absolute 5
Reference plane absolute 0
Safety distance 2
Final drilling absolut .... -30
Depth incr. 0
First drilling depth .... -10
First depth 0
Degression 3
Dwell at drilling depth 0
Dwell at starting time 0
Feedrate factor 1
Machining type 0
Tool axis 1
Minimum drilling depth 1
Variable return path 0
Dwell time at final drilling depth 0
Lead distance 0

```
G54
TRANS Z70
(G17)**
T8 D1
G95 S1000 M3 F0.12
G0 X0 Z5
Cycle 83 (5, 0, 2, -30, 0, -10, 0, 3, 0, 0, 1, 0,1,1,0,0,0)
G0 X100 Z10
(G18)**
M30
```

**...only at cycle version 4

EMCO

Drilling with driven tools (axial)

Return plane, absolute 5
Reference plane absolute 0
Safety distance 2
Final drilling absolut .... -30
Depth incr. 0
First drilling depth .... -10
First depth 0
Degression 3
Dwell at drilling depth 0
Dwell at starting time 0
Feedrate factor 1
Machining type 0
Tool axis 1
Minimum drilling depth 1
Variable return path 0
Dwell time at final drilling depth 0
Lead distance 0

Drilling with driven tools (radial)

Return plane, absolute 47
Reference plane absolute 45
Safety distance 2
Final drilling depth absolute 5
Depth incr. 0
First drilling depth 25
First depth 0
Degression 3
Dwell at drilling depth 0
Dwell at starting time 0
Feedrate factor 1
Machining type 0
Tool axis 2
Minimum drilling depth 1
Variable return path 0
Dwell time at final drilling depth 0
Lead distance 0
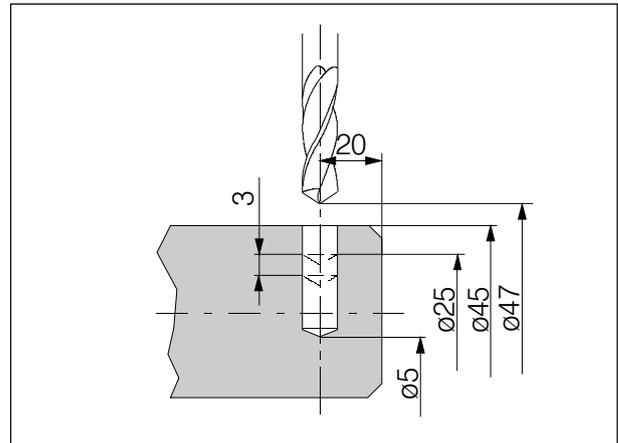
G54
TRANS Z70
(G17)**
T7 D1
SPOS[1] =0
SETMS(2)
G95 S1000 M3 F0.12
G0 X20 Z5
Cycle 83 (5, 0, 2, -30, 0, -10, 3, 0, 0, 1, 0,1,1,0,0,0)
G0 X50 Z20
M5
SETMS(1)
(G18)**
M30

G54
TRANS Z70
(G19)**
T5 D1
SPOS[1] =0
SETMS(2)
G95 S1000 M3 F0.12
G0 X47 Z-20
Cycle 83 (47, 45, 2, 5, 0, 25, 0, 3, 0, 0, 1, 0,2,1,0,0,0)
G0 X50 Z20
M5
SETMS(1)
(G18)**
M30

CYCLE83E Deep hole drilling

This cycle is used for drilling deep holes, either in x - axis or z- axis direction.

> **Advantage:**
> * No plane selection (only at cycle version 4).
> * Drilling direction can be programmed directly.
> * Tool type 500 can be used

Drilling on the main spindle
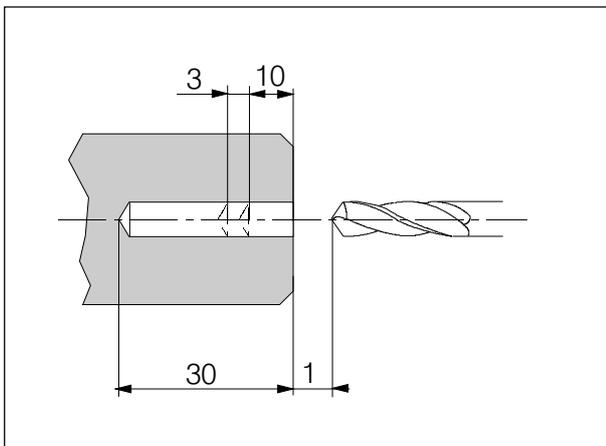
Reference plane absolute 1
Final drilling absolute .... -30
First drilling depth absolute .... -10
Degression 3
Dwell at drilling depth 0
Dwell at starting time 0
0 = chipbreaking; 1 = stock removal 1
0 = X - direction; 1 = Z - direction 1

```
G54
TRANS Z70
T7 D1
G95 S1000 M3 F0,12;
G0 X0 Z2
CYCLE83E(1,-75,-30,10,0,0,1,1)
G0 X100 Z10
M30
```

Drilling with driven tools (axial)

Reference plane absolute 1
Final drilling absolute .... -30
First drilling depth absolute .... -15
Degression 3
Dwell at drilling depth 0
Dwell at starting time 0
0 = chipbreaking; 1 = stock removal 1
0 = X - direction; 1 = Z - direction 1

Drilling with driven tools (radial)

Reference plane absolute 62
Final drilling absolute .... -5
First drilling depth absolute 40
Degression 5
Dwell at drilling depth 0
Dwell at starting time 0
0 = chipbreaking; 1 = stock removal 1
0 = X - direction; 1 = Z - direction 0
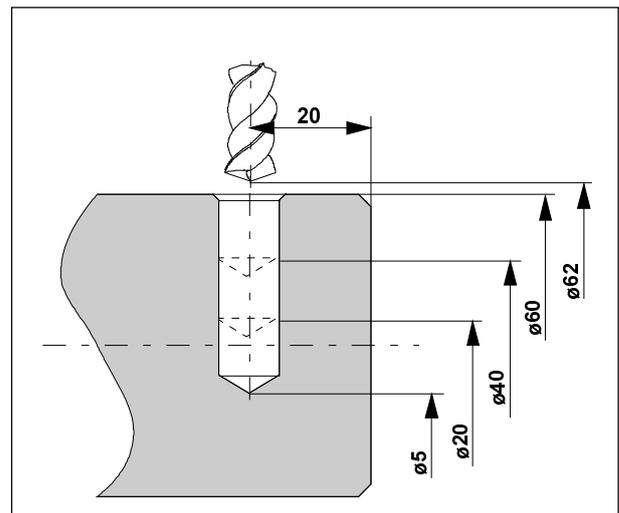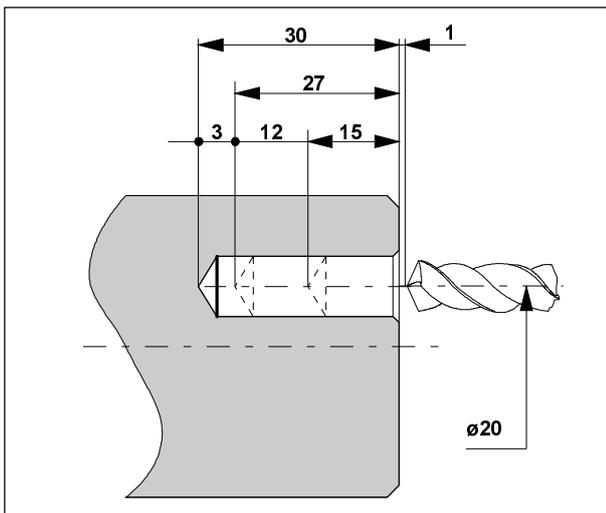

```
G54
TRANS Z70
T7 D1 ;
SPOS[1]=0 ;
SETMS(2) ;
G95 S1000 M3 G0 X20 Z2 F0,12
G0 X20 Z5
CYCLE83E(1,-30,-15,3,0,0,1,1)
G0 X100 Z10 M5
SETMS(1) ;
M30
```

```
G54
TRANS Z70
T5 D1 ;
SPOS[1]=0 ;
SETMS(2) ;
G95 S1000 M3 G0 X70 Z-20 F0,12
G0 X62 Z-20
CYCLE83E(62,5,40,-5,0,0,1,0)
G0 X50 Z10 M5
SETMS(1) ;
M30
```

CYCLE84 Rigid tapping

(Only for PC-Turn 155 and CT 155, CT 450)

CYCLE84 (RTP,RFP,SDIS,DP,DPR,DTB,SDAC,MPIT,PIT,POSS,
        SST,SST1,AXN,PTAB,TECH,VARI,DAM,VRT)

Parameter additional to Cycle81:

SDAC   spindle direction after cycle end   **S**pindle **D**irection **A**fter **C**ycle
        3: right, 4: left, 5: spindle stop

MPIT   thread pitch as nominal value   **M**etrical **PIT**ch
        Thread pitch for regular metric thread, value range 3 (M3) - 48 (M48).

PIT   thread pitch in mm   **PIT**ch
        Value range 0,001 - 2000 mm.

        (with sign for the rotation direction)

        Program either MPIT or PIT. Contradictious values trigger an alarm.

POSS   spindle position for exact stop   **POS**ition of **S**pindle
        Before the cycle the spindle will be positioned with POSS.

SST   spindle speed for tapping   **S**pindle **S**peed for **T**apping

SST1   spindle speed for retraction   **S**pindle **S**peed for **T**apping

AXN(*) Tool axis

        Selection of the tool axis:
        X=2; Z=1

PTAB(*) Evaluatioon of the threading pitch PIT

        0: correspondingly programmed measuring system inch/metric

        1: pitch in mm

        2: pitch in threads per inch

        3: pitch in inch/revolution

TECH(*) Technological settings

        UNITS DIGIT: exact stop

        0: programmed as before a cycle call

        1: (G601)

        2: (G602)

        3: (G603)

        TENS DIGIT: pilot control

        0: programmed as before a cycle call

        1: with pilot control (FFWON)

        2: without pilot control (FFWOFF)

        HUNDREDS DIGIT: acceleration

        0: programmed as before a cycle call

        1: jerk-limited acceleration of the axes

        2: rapid acceleration of the axes

        3: reduced acceleration of the axes

        THOUSANDS PLACE:

        0: activate spindle operation again (for MCALL)

        1: remain in position-controlled operation (for MCALL)

VARI(*) Machining mode

    0: continuous tapping

    1: deep-hole tapping with chip breaking

    2: deep-hole tapping with chip removal

DAM(*) Incremental drilling depth

    without arithmetic sign

VRT(*) Variable retraction path for chip breaking

    without arithmetic sign
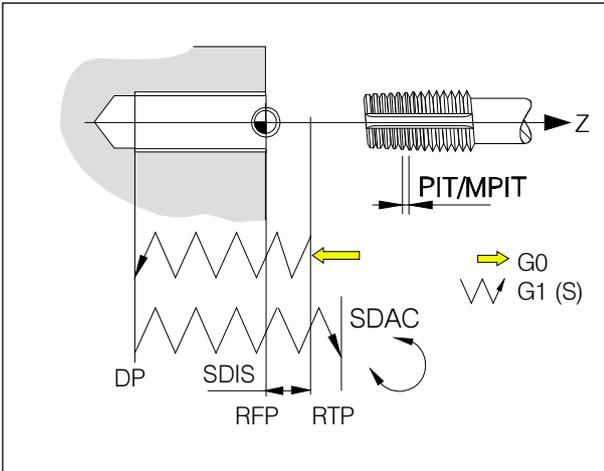
Machining sequence:

- Previous to the cycle the tool must be placed over the hole position (X=0).

- Rapid traverse to safety distance.

- Oriented spindle stop POSS.

- Tapping to end depth DP with spindle speed SST, spindle rotation and feed are synchronized.

- Dwell time at end depth.

- Spindle direction change.

- Retraction to safety distance with spindle speed SST1.

- Rapid retraction to retraction plane RTP.

- Establishing the spindle direction SDAC.

Drilling on the main spindle
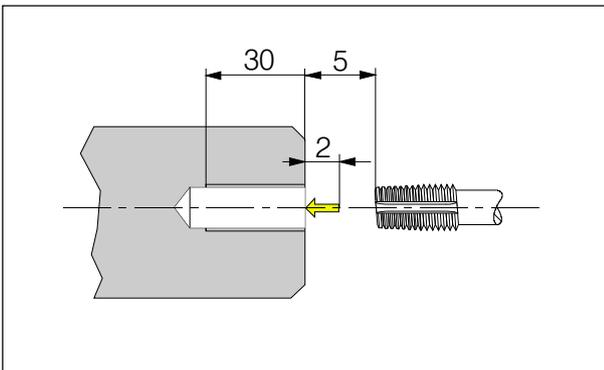
Return plane, absolute 5
Reference plane absolute 0
Safety distance.. 2
Final drilling depth ....
-30
Depth incr. 0
Dwell time 0
Direction of rotation after end of cycle 3
Thread lead as thread size 0
Thread lead as value.. 1
Spindle position 0
Speed for tapping 100
Speed for retract 100
Tool axis 1
Evaluation of the thread pitch............................. 0
Technological settings 0
Machining mode 0
Incremental drilling depth 0
Variable return path 0

```
G54
TRANS Z70
(G17)**
T8 D1
G0 X0 Z5
Cycle 84 (5, 0, 2, -30, 0, 0,  3, 0, 1, 0, 100, 100,
1,0,0,0,0,0)
G0 X100 Z80
(G18)**
M30
```

Note:
PC Turn 155, CT 450:
Spindle speed in the Rigid Tapping cycle max. 100U/min.

**...only at cycle version 4

EMCO

Drilling with driven tools without compensation chuck (axial)

Drilling with driven tools without compensation chuck (radial)

It can be worked with normal drilling and milling holders for axial or radial threads

	(axial)	(radial)
Return plane, absolute	5	47
Reference plane absolute	0	45
Safety distance	2	2
Final drilling depth	-20	25
Depth incr.	0	0
Dwell time	0	0
Direction of rotation after end of cycle	3	3
Thread lead as thread size	0	0
Thread lead as value	1	1,5
Spindle position	0	0
Speed for tapping	600	800
Speed for retract	800	1000
Tool axis	1	2
Evaluation of the thread pitch	0	0
Technological settings	0	0
Machining mode	0	0
Incremental drilling depth	0	0
Variable retraction path	0	0


```
G54
TRANS Z70
T7 D1
SPOS[1]=0
SETMS(2)
G0 X0 Z5
Cycle 84 (5, 0, 2, -20, 0, 0,  3, 0, 1, 0, 600,
800,1,0,0,0,0,0)
SETMS(1)
M30
```

```
G54
TRANS Z70
T5 D1
SPOS[1]=0
SETMS(2)
G0 X47 Z-20
Cycle 84 (47, 45, 2, 25, 0, 0,  3, 0, 1.5, 0, 800,
1000,2,0,0,0,0,0)
SETMS(1)
M30
```

EMCO

CYCLE84E Deephole drilling
only CT 155, CT 450

Advantage:
- No plane selection (only at cycle version 4).
- Drilling direction can be programmed.
- Tool type 500 can be used

Thread M10×1,5

20
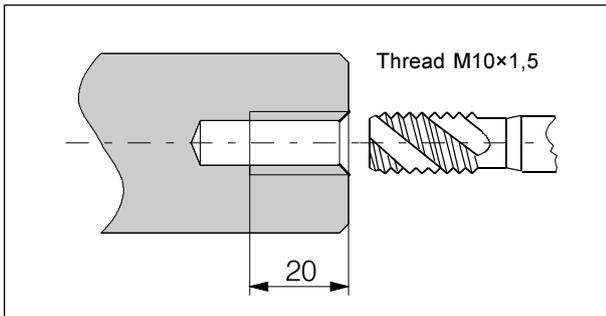
Drilling on the main spindle

Reference plane absolute 3
Final drilling depth .... -25
Thread lead as value
(with sign for the rotation direction) 1,5
Speed for tapping 100
Speed for retract 100
0 = X - axis; 1 = Z - axis 1

```
G54
TRANS Z70
T7 D1 M5
G0 X0 Z5
CYCLE84E(3,-25,1.5,100,100,1)
G0 X50 Z10
M30
```

Note:
PC Turn 155, CT 450:
Spindle speed in the Rigid Tapping cycle max. 100U/min.

EMCO

Drilling with driven tools without compensation chuck (axial)

Drilling with driven tools without compensation chuck (radial)

It can be worked with normal drilling and milling holders for axial or radial threads

Thread M6×1

ø20

Thread M10×1,5

ø60

ø40
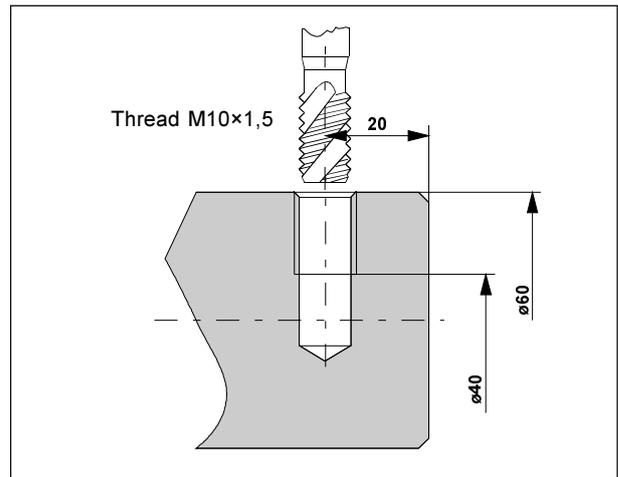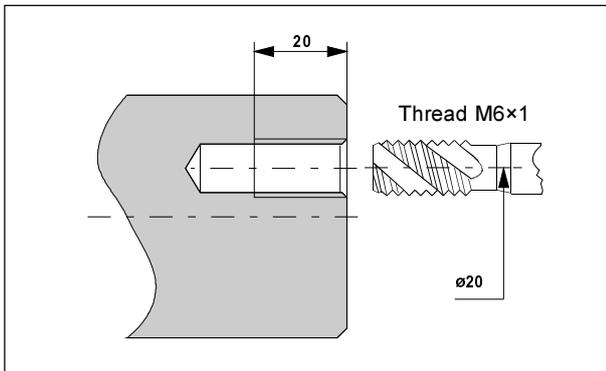
Reference plane absolute	2
Final drilling depth	-20
Thread lead as value (with sign for the rotation direction)	1
Speed for tapping	600
Speed for retract	800
0 = X - axis; 1 = Z - axis	1

Reference plane absolute	65
Final drilling depth	40
Thread lead as value (with sign for the rotation direction)	1,5
Speed for tapping	600
Speed for retract	800
0 = X - axis; 1 = Z - axis	0

```
G54
TRANS Z70
T7 D1 ;
SPOS[1]=0
SETMS(2)
G0 X20 Z2
CYCLE84E(2,-20,1,600,800,1)
G0 X50 Z10 M5
SETMS(1)
M5
M30
```

```
G54
TRANS Z70
T5 D1
SPOS[1]=0
SETMS(2) ;
G0 X70 Z-20
CYCLE84E(65,40,1.5,600,800,0)
G0 X80 Z10 M5
SETMS(1)
M5
M30
```

EMCO

CYCLE840 Tapping with compensation chuck

CYCLE840 (RTP,RFP,SDIS,DP,DPR,DTB,SDR,SDAC,ENC,
            MPIT,PIT,AXN,PTAB,TECH)

Parameter additional to Cycle81:

SDR   spindle direction for retraction   **S**pindle **D**irection for **R**etraction
        0: automatic change, 3: right, 4: left

SDAC   spindle direction after cycle end   **S**pindle **D**irection **A**fter **C**ycle
        3: right, 4: left, 5: spindle stop

ENC   use encoder   **ENC**oder
        0: use encoder, 1: do not use encoder, on machines without encoder this
        parameter will be ignored

MPIT   thread pitch as nominal value   **M**etrical **PIT**ch
        Thread pitch for regular metric thread, value range 3 (M3) - 48 (M48).

PIT   thread pitch in mm   **PIT**ch
        Value range 0,001 - 2000 mm.

        Program either MPIT or PIT. Contradictious values trigger an alarm.

AXN(*) Tool axis
        Selection of tool axis:
        X=2; Z=1

PTAB(*) Evaluation of the threading pitch PIT
        0: correspondingly programmed measuring system inch/metric
        1: pitch in mm
        2: pitch in threads per inch
        3: pitch in inch/revolution

TECH(*) Technological settings
        UNITS DIGIT: exact stop
        0: programmed as before a cycle call
        1: (G601)
        2: (G602)
        3: (G603)
        TENS DIGIT: pilot control
        0: programmed as before a cycle call
        1: with pilot control (FFWON)
        2: without pilot control (FFWOFF)
        HUNDREDS DIGIT: brake actuation point
        0: without calculation
        1: with calculation
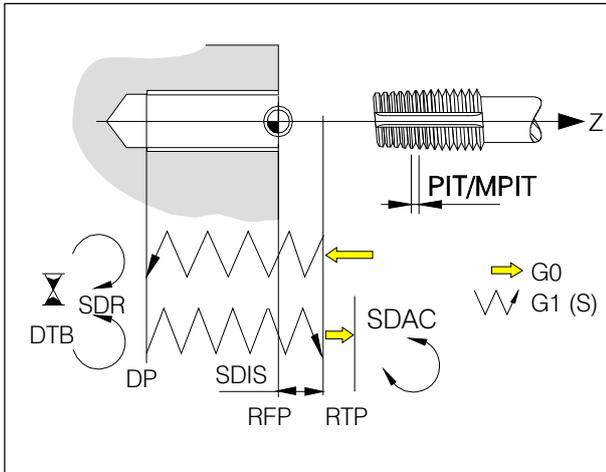
EMCO

Machining sequence:

- Previous to the cycle the tool must be placed over the hole position (X=0).

- Rapid traverse to safety distance.

- Tapping to end depth DP with programmed spindle speed.

- Dwell time at end depth.

- Change of spindle direction according SDR.

- Retraction to safety distance

- Rapid retraction to retraction plane RTP.

- Establishing the spindle direction SDAC.

Drilling on the main spindle

Return plane, absolute 5
Reference plane absolute 0
Safety distance 2
Final drilling depth ....-20
Depth incr. 0
Dwell time 0
Direction of rotation for retract 4
Direction of rotation after cycle end 3
0: with encoder, 1: without encoder 0
Thread lead as thread size 0
Thread lead as value.. 1
Tool axis 1
Evaluation of thread pitch.................................. 0
Technological settings 0
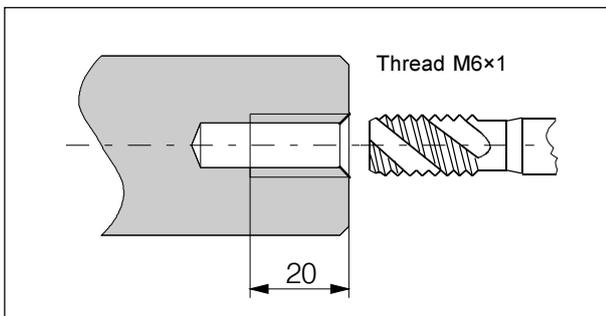
```
G54
TRANS Z70
(G17)**
T8 D1
G97 S600 M3
G0 X0 Z5
Cycle 840 (5, 0, 2, -20, 0, 0,  4, 3, 0, 0, 1,1,0,0)
G0 X100 Z80
(G18)**
M30
```

Thread M6×1

20

**...only at cycle version 4

EMCO

Drilling with driven tools with compensation chuck

Return plane, absolute 5
Reference plane absolute 0
Safety distance 2
Final drilling depth .... -20
Depth incr. 0
Dwell time 0
Direction of rotation for retract 4
Direction of rotation after cycle end 3
0: with encoder, 1: without encoder 1
Thread lead as thread size 0
Thread lead as value 1
Tool axis 1
Evaluation of the thread pitch 0
Technological settings 0

G54
TRANS Z70
(G17)**
T7 D1
SPOS[1]=0
SETMS(2)
G97 S800 M3
G94 F800*
G0 X20 Z5
Cycle 840 (5, 0, 2, -20, 0, 0, 4, 3, 1, 0, 1,1,0,0)
G0 X80 Z80
M5
SETMS(1)
(G18)**
M30

Drilling with driven tools with compensation chuck (radial)

Return plane, absolute 47
Reference plane absolute 45
Safety distance 2
Final drilling depth 25
Depth incr. 0
Dwell time 0
Direction of rotation for retract 4
Direction of rotation after cycle end 3
0: with encoder, 1: without encoder 1
Thread lead as thread size 0
Thread lead as value 1
Tool axis 2
Evaluation of the thread pitch 0
Technological settings 0
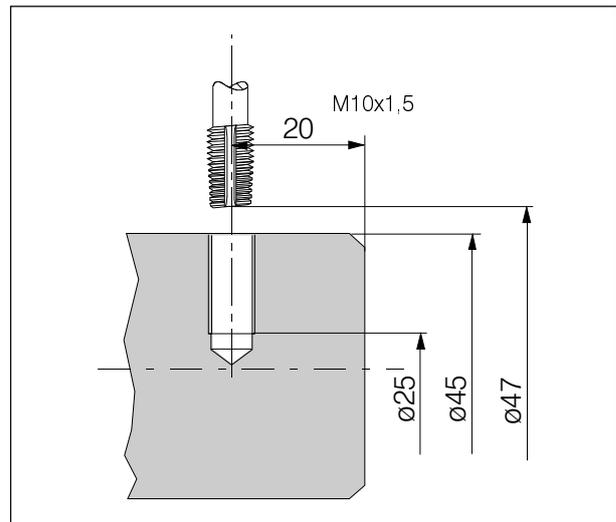
G54
TRANS Z70
(G19)**
T5 D1
SPOS[1]=0
SETMS(2)
G97 S800 M3
G94 F800*
G0 X47 Z-20
Cycle 840 (47, 45, 2, 25, 0, 0, 4, 3, 1, 0, 1,2,0,0)
G0 X50 Z80
M5
SETMS(1)
(G18)**
M30

* In case you work with a spindle encoder (ENC=0), the feed data is not necessary for CT 155 and CT 450.
(Otherwise feed [F] = spindle speed [n] x pitch [p])

EMCO

CYCLE85 Boring 1, CYCLE89 Boring 5

CYCLE85 (RTP,RFP,SDIS,DP,DPR,DTB,FFR,RFF)
CYCLE89 (RTP,RFP,SDIS,DP,DPR,DTB)

Boring 1 and 5 work similar CYCLE82.

Differences to CYCLE82:

• The infeed feed rate is not the last programmed F value but will be programmed with parameter FFR in the cycle call.
• The retraction feed is not rapid feed but will be programmed with parameter RFF in the cycle call.

| FFR | infeed feed rate | **F**orward **F**eed **R**ate |
| RFF | retraction feed rate | **R**etraction **F**eed |

CYCLE86 Boring 2

CYCLE86 (RTP,RFP,SDIS,DP,DPR,DTB,SDIR,RPA,RPO,RPAP,POSS)

Boring 2 works similar CYCLE82.
Only a boring head tool is allowed.

Differences to CYCLE82:

- The rotational direction is programmed in the cycle with SDIR.
- At the boring ground is an oriented spindle stop (POSS) and the boring head can be lifted from the surface with RPA, RPO, RPAP in X/Y/Z to avoid scratching the surface while retraction.

SDIR	spindle direction 3: right, 4: left	**S**pindle **DIR**ection
RPA	lift off movement in X incremental with sign	**R**etraction **P**osition **A**bscissa
RPO	lift off movement in Y incremental with sign	**R**etraction **P**osition **O**rdinate
RPAP	lift off movement in Z incremental with sign	**R**etraction **P**osition **AP**plicate
POSS	spindle position for exact stop	**POS**ition of **S**pindle
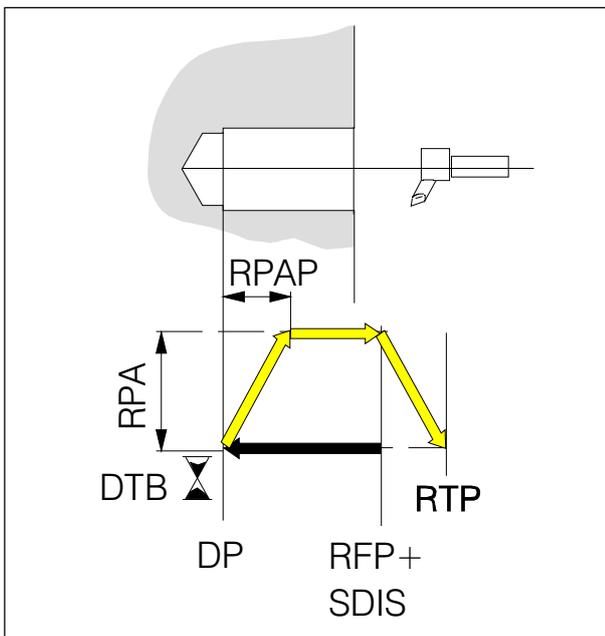
The lift off movement must occur in opposite direction to the boring head cutter edge.

The lift off path must be smaller than the outstanding length of the cutter edge out of the boring head.

CYCLE87 Boring 3

CYCLE87 (RTP,RFP,SDIS,DP,DPR,SDIR)

CAUTION: Boring with program stop M0 at hole ground,
Retraction is done after pressing NC Start without spindle rotation.

CYCLE88 Boring 4

CYCLE87 (RTP,RFP,SDIS,DP,DPR,DTB,SDIR)

CAUTION: Boring with dwell time and program stop M0 at hole ground,
Retraction is done after pressing NC Start without spindle rotation.

Turning Cycles

Cycle 93	Grooving cycle
Cycle 94	Undercut cycle
Cycle 95	Stock removal cycle
Cycle 96	Thread undercut
Cycle 97	Thread cutting cycle
Cycle 98	Chaining of threads

EMCO

CYCLE 93 Grooving cycle

CYCLE93 (SPD,SPL,WIDG,DIAG,STA1,ANG1,ANG2,RCO1,RCO2,
          RC11,RC12,FAL1,FAL2,IDEP,DTB,VARI)

SPD	start point in X without sign	**S**tart **P**oint **D**iameter
SPL	start point in Z	**S**tart **P**oint **L**ength
WIDG	groove width at groove ground without sign	**WID**th **G**round
	must be larger or equal tool width	

DIAG   groove depth related to start point
         without sign.

STA1   angle between contour and turning axis
         $0<=STA1<=180°$.

ANG1   flank angle 1   **ANG**le 1
         Flank angle at the side of the start point without sign
         $0<=ANG1<=89.999°$

ANG2   flank angle 2   **ANG**le 2
         Flank angle at the side opposite to the start point without sign
         $0<=ANG2<=89.999°$

RCO1   radius / chamfer 1 outside   **R**adius **C**orner **O**utside 1
         Radius /chamfer at the outside corner on the side of the start point
         Radius: positive sign, chamfer: negative sign

RCO2   Radius/Fase 2 outside   **R**adius **C**orner **O**utside 2
          Radius /chamfer at the outside corner on the side opposite to the start point
         Radius: positive sign, chamfer: negative sign

RCI1   Radius/Fase 1 inside   **R**adius **C**orner **I**nside 1
          Radius /chamfer at the inside corner on the side of the start point
         Radius: positive sign, chamfer: negative sign

RCI2   Radius/Fase 2 inside   **R**adius **C**orner **I**nside 2
          Radius /chamfer at the inside corner on the side opposite to the start point
         Radius: positive sign, chamfer: negative sign

FAL1	finishing allowance at groove ground	**F**inishing **AL**lowance 1
FAL2	finishing allowance at the flanks	**F**inishing **AL**lowance 2

         If FAL1 and FAL2 are programmed, roughing leaves back these allowances.
         Afterwards a countour-parallel cut occurs along the final contour with the
         same tool.

IDEP   infeed depth without sign   **I**nfeed **DEP**th

DTB   dwell time at groove ground in [s]   **D**well **T**ime at end de**P**th
         The tool lifts off the groove ground after the programmed dwell time.

VARI   machining variant   **Vari**ant

VRT[*)]   Variable return distance from contour

Note:
Both cutting edges must be measured at the cut-in tool. The values for the two edges must be entered in subsequent D numbers.
E.g. if T2 D1 is called at tool for the grooving cycle, the second edge4 must be entered in D2.
The cycle itself selects the corresponding tool correction fo each machining step of the cycle and activates it automatically.

VARI=x1

VARI=x2

VARI=x3

VARI=x4

VARI=x5

VARI=x6

VARI=x7

VARI=x8

Variant of groove

VARI=0x

VARI=1x

Chamfer calculation
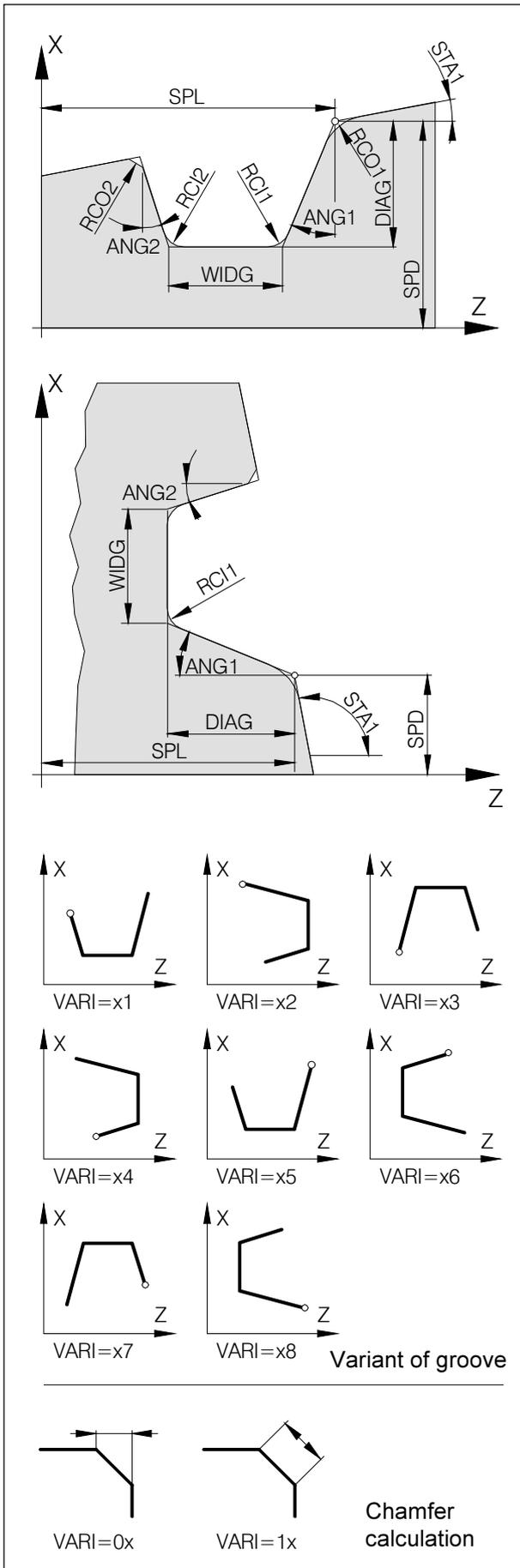
SPD, SPL
These coordinates define the start point of the groove in X (SPD) and Z (SPL).

WIDG, DIAG
Groove width (WIDG) and groove depth (DIAG), related to the start point.
When the groove is larger in width than the tool, the whole width will be worked off in several steps with even overlapping.

STA1
Angle of the contour on which the groove will be produced. This angle is related to the Z axis.
This angle will be indicated for that the lateral infeed outside can occur parallel to the contour.

ANG1, ANG2
By separated flank angles, asymmetric groves can be produced.

RCO1, RCO2, RCI1, RCI2
Radius / chamfer at the 4 corners of the groove.
Radius: positive sign
Chamfer: negative sign
The variant of chamfer calculation is determined by the parameter VARI.

FAL1, FAL2
For groove ground (FAL1) and flanks (FAL2) different finishing allowances can be determined.
Roughing leaves back these allowances. Afterwards a countour-parallel cut occurs along the final contour with the same tool

IDEP
Infeed depth. The axis-parallel dive-in is divided in several depth infeeds. After every infeed the tool will be retracted for 1 mm to break the chips.
After each infeed the tool is retracted for chip breaking by the distance that was programmed under VRT.
The parameter IDEP must be programmed anyhow.

DTB
The dwell time must be as long as at least one spindle revolution lasts.

VARI
The ten's place digit (**x**x) determines how the chamfer length will be calculated.
The one's place digit (x**x**) defines the kind (= position) of the groove.

VRT
Return distance during grooving. When VRT=0 (parameter not programmed) the tool is retracted by 1 mm.

EMCO

Example CYCLE93 Groove

Longitudinal groove outside.
The start point is right at X70 Z60.
The cycle uses the tool corrections D1 and D2 of the
tool T1.
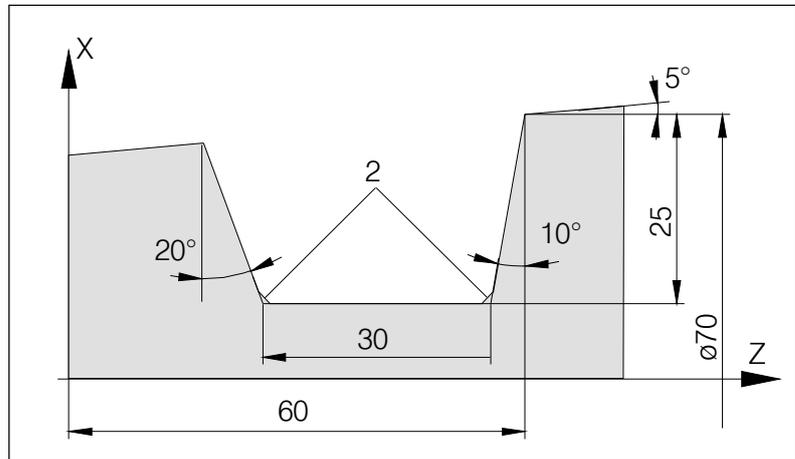
Cycle parameter:
CYCLE93   (SPD,SPL,WIDG,DIAG,STA1,ANG1,ANG2,RCO1,RCO2,RC11,RC12,FAL1,FAL2,IDEP,DTB,VARI,VRT)

1.	Start point in X	70	mm
2.	Start point in Z	60	mm
3.	Groove width:	30	mm
4.	Groove depth:	25	mm
5.	Angle contour - longitudinal axis	5	°
6.	Flank angle start point:	10	°
7.	Flank angle opposite:	20	°
8.	Radius / chamfer outside start point:0		
9.	Radius / chamfer outside opposite:	0	
10.	**Chamfer** inside start point:	-2	mm
11.	**Chamfer** inside opposite:	-2	mm
12.	Finishing allowance ground:	0,2	mm
13.	Finishing allowance flank:	0,1	mm
14.	Infeed depth:	10	mm
15.	Dwell time:	1	s
16.	Variant:	05	
17.	Variable retraction path:	1	mm

Programm:

N10   G0   X90   Z65   T1   D1   S400 M3   Beginning point before cycle

N20   G95   F0.2   Technological data

N30   CYCLE93(70,60,30,25,5,10,20,0,0,-2,-2,0.2,0.11,10,1,5, 1)   Cycle call

N40   G0   X50   Z65   Next position

...

Example CYCLE93 Groove

Face groove outside.
The start point is top at X40 Z0.
The cycle uses the tool corrections D1 and D2 of the
tool T9.
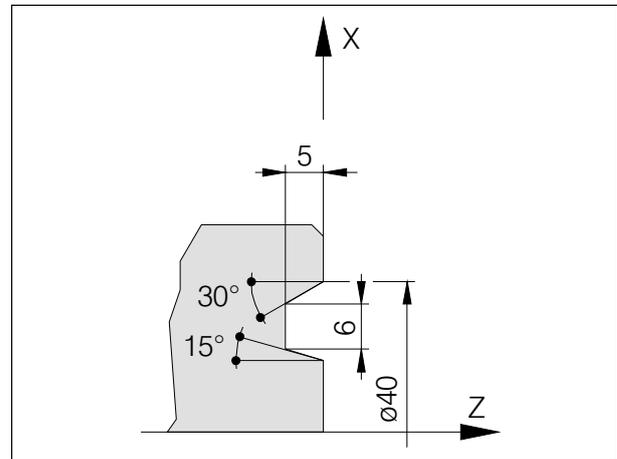
Cycle parameter:
CYCLE93   (SPD,SPL,WIDG,DIAG,STA1,ANG1,ANG2,RCO1,RCO2,RC11,RC12,FAL1,FAL2,IDEP,DTB,VARI,VRT)

1. Start point in X	40	mm
2. Start point in Z	0	mm
3. Groove width:	6	mm
4. Groove depth:	5	mm
5. Angle contour - longitudinal axis	90	°
6. Flank angle start point:	30	°
7. Flank angle opposite:	15	°
8. Radius / chamfer outside start point:	0	
9. Radius / chamfer outside opposite:	0	
10. Radius / chamfer inside start point:	0	
11. Radius / chamfer inside opposite:	0	
12. Finishing allowance ground:	0,2	mm
13. Finishing allowance flank:	0,1	mm
14. Infeed depth:	5	mm
15. Dwell time:	1	s
16. Variant:	16	
17. Variable retraction path:	1	mm

Programm:

N10	G0	X200 Z100					Beginning point before cycle
N20	T9	D1	G96	S180	M4	F0.12	Technological data
N30	CYCLE93(40,0,6,5,90,30,15,0,0,0,0,0.2,0.1,5,1,16,1)						Cycle call
N40	G0	X200 Z100					Next position

...

CYCLE 94 Undercut cycle

CYCLE94 (SPD,SPL,FORM,VARI)

SPD   start point in X without sign   **S**tart **P**oint **D**iameter
SPL   start point in Z   **S**tart **P**oint **L**ength
FORM   form of undercut   **FORM**
            Values:
            E: for Form E according DIN 509
            F: for Form F according DIN 509
VARI[*] Determination of the undercut position **VARI**ante
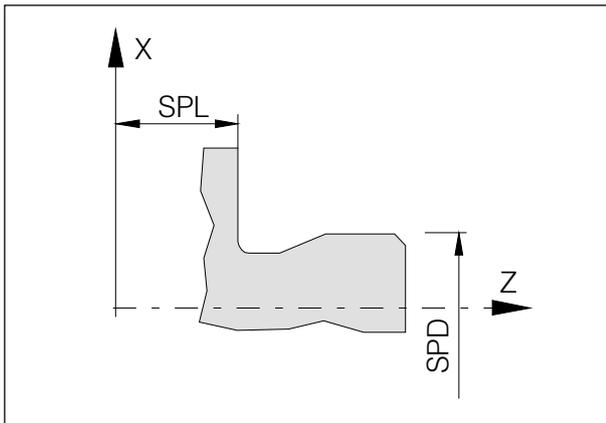
This cycle produces undercuts according DIN 509 of the form E and F with common strain for finished part diameters > 3 mm.
Thread undercuts (Form A and D DIN 76) see CYCLE 96.

SPD, SPL
SPD defines the finished part diameter of the undercut.
Finished part diameters below 3 mm can not be produced with this cycle.
SPL defines the finish dimesion (shoulder) in Z.
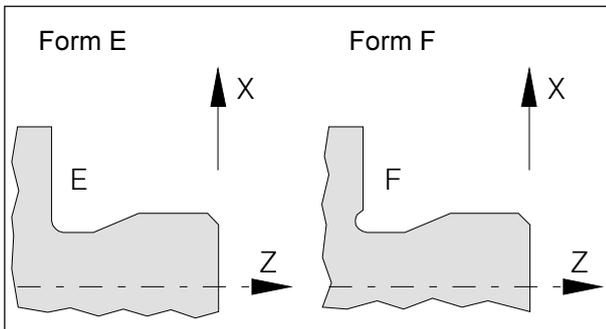
FORM
Form of the undercut according DIN 509.
Form E: for workpieces with one machined surface (circumference).
Form F: for workpieces with two machined rectangular aligned surfaces (shoulder and circumference).
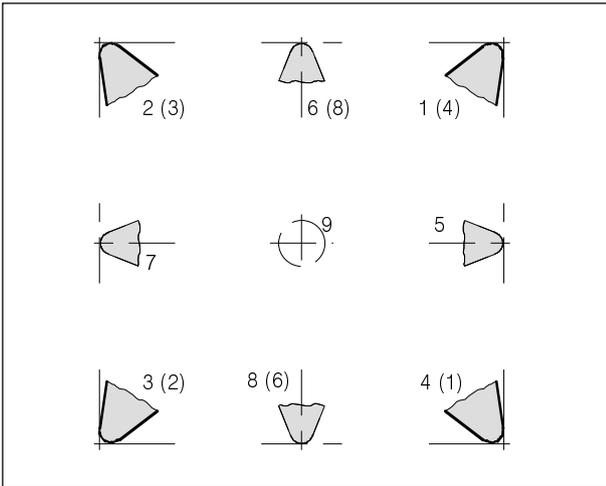
EMCO

For machines with the tool below (in front of) the turning axis (e.g. PC TURN 50/55), the values in brackets are valid.

VARI:
Only tools with the cutter positions 1, 2, 3, 4 can be used for this cycle.

When a clearance angle is entered in the tool data, it will be monitored.
After detecting that the form of the undercut can not be produced with the selected toolbecause of a too small clearance angle, the message: "changed form of undercut" will appear at the screen.
Machining will be continued (the error in form normally is very small).

EMCO

CYCLE 95 Stock removal cycle

CYCLE95 (NPP,MID,FALZ,FAX,FAL,FF1,FF2,FF3,VARI,DT,DAM,VRT)

NPP	name of the subprogram	**N**ame **P**art **P**rogram
MID	maximum infeed depth without sign	**M**aximum **I**nfeed **D**epth
FALZ	finishing allowance in Z without sign	**F**inishing **AL**lowance **Z**
FALX	finishing allowance in X without sign	**F**inishing **AL**lowance **X**
FAL	finishing allowance parallel to contour without sign	**F**inishing **AL**lowance
FF1	feed for roughing cuts without undercut	
FF2	feed for roughing - dive-in in undercuts	
FF3	feed for finishing	
VARI	machining variant 1..12	**Vari**ant

HUNDREDS DIGIT:

0: with retracing the contour

> No residual edges will remain, the contour is retraced in an overlapping way. This means that the contour is retraced at several intersection points.

2: without retracing the contour

> It is only retraced up to the previous roughing point, then the tool is lifted. Depending on the ratio of the tool radius to the cutting depth (MID), there may remain residual edges.

DT	dwell time for chip break while roughing
DAM	traverse path after that every roughing cut will be interrupted for chip breaking
VRT[(*)]	Set-up clearance from the contour during roughing without arithmetic sign

Function:
The stock removal cycle works off a contour that is stored in a subprogram.
The contour can be machined outside or inside, longitudinal or face side.
The contour can be roughed, finished or complete machined.

Tool position before cycle:
The last position before cycle call must be approached with G40 (cutter radius compensation off).
Outside machining: Before cycle call the tool must be outside the greatest diameter of the contour subprogram.
Inside machining: Before cycle call the tool must be inside the smallest diameter of the contour subprogram.

EMCO
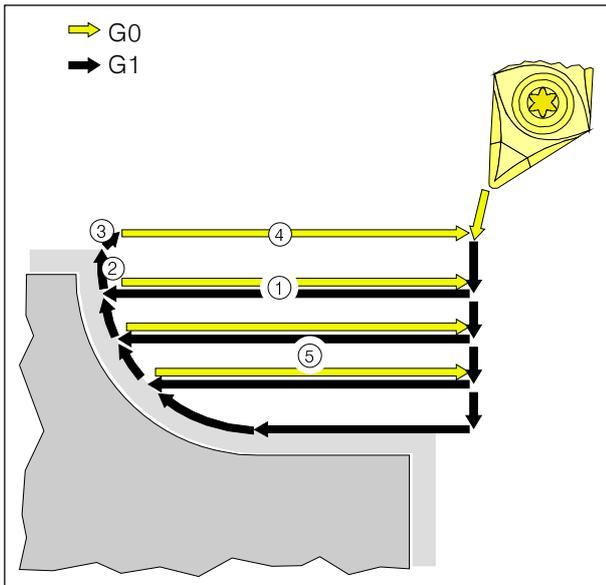
Machining sequence:

Roughing without undercut elements

• The roughing cuts are done with G1 and feed rate FF1.
• The roughing cuts occur parallel to the axis until finishing allowance (1), afterwards parallel to the contour (2).
• After every roughing cut is a lift off in X and Z for tool radius + 1 mm (3) and retraction with G0 (4).
• This sequence is repeated until the end depth (with finishing allowance) is reached (5).
  The infeed depths are divided evenly, for that they are smaller / equal the programmed parameter MID.
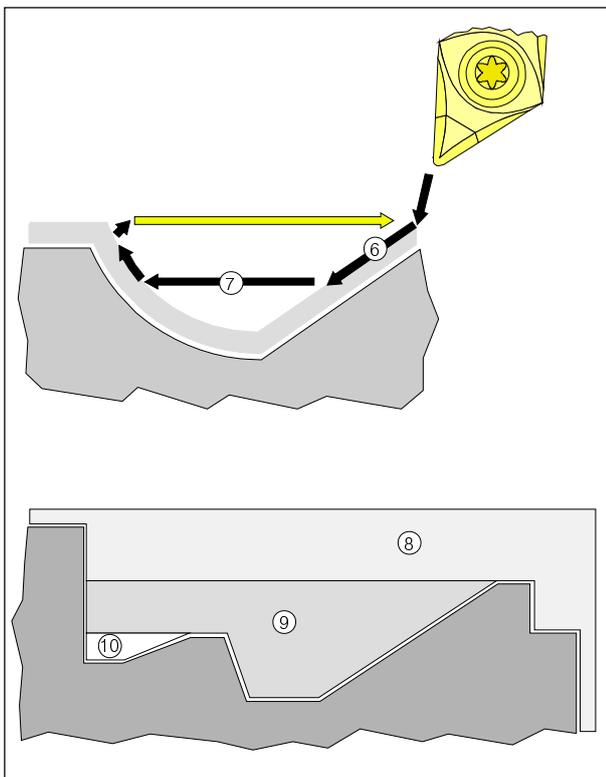
Roughing the undercut elements

• The infeed at the undercut occurs parallel to the contour (6) with G1 and feed rate FF2.
• The roughing cuts parallel to the axis within the undercut area (7) occur with G1 and feed rate FF1.
• Sequence of roughing:
  - Roughing without undercut (8)
  - Roughing 1st undercut (9)
  - Roughing 2nd undercut (10) etc.

Finishing

• The contour start point will be approached in both axes simultanuously.
• Finishing occurs along the contour with G1, G2, G3 and with feed rate FF3.
• Retraction occurs with G0.

NPP
This parameter is the name of the contour subprogram. The name must be in quotation marks, e.g. "CONT1".

MID
Maximum infeed depth for roughing.
The total depth for roughing will be divided evenly in several single infeeds. These infeed depths are divided evenly, for that they are smaller / equal the programmed parameter MID.
Example:
Total depth = 19 mm, MID = 4 mm
-> 5 infeeds with each 3,8 mm will be machined.

FALZ, FALX, FAL
Finishing allowance for roughing
FALZ   Finishing allowance in Z
FALX   Finishing allowance in X
FAL   Finishing allowance parallel to the contour
It is not useful to program all 3 parameter (the values will be aaded).
Program either the values for FALZ and FALX and 0 for FAL or vice versa.
When no finishing allowance is programmed, roughing is proceeded until final contour.

FF1, FF2, FF3
Feed rates for the different maching steps:
FF1   Roughing
FF2   Roughing - dive-in in undercuts
FF3   Finishing.

EMCO

VARI
VARI defines the kind of machining (roughing, finishing, complete), the direction of machining (longitudinal or face) and and the side of machining (inside or outside).
HUNDREDS DIGIT:
0: with retracing the contour
2: without retracing the contour
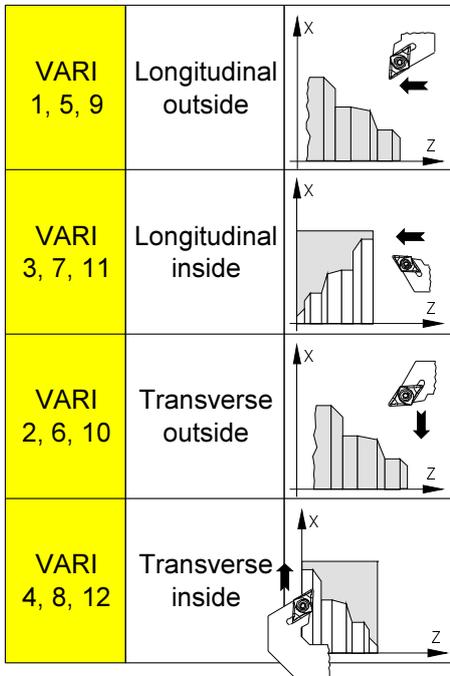
VARI	Longitudinal / Transverse	Outside / Inside	Type of machining
1	L	Outside	Roughing
2	T	Outside	Roughing
3	L	Inside	Roughing
4	T	Inside	Roughing
5	L	Outside	Finishing
6	T	Outside	Finishing
7	L	Inside	Finishing
8	T	Inside	Finishing
9	L	Outside	Complete machining
10	T	Outside	Complete machining
11	L	Inside	Complete machining
12	T	Inside	Complete machining

VARI 1, 5, 9	Longitudinal outside	
VARI 3, 7, 11	Longitudinal inside	
VARI 2, 6, 10	Transverse outside	
VARI 4, 8, 12	Transverse inside	

DT, DAM
These parameter interrupt the axis-parallel movement while roughing to break the chip.
DT   dwell time
DAM   traverse path after that the movement should be stopped
Programming DAM=0 means no interruption, the dwell time will not be executed.

VRT (set-up clearance)
When VRT=0 (parameter not programmed) the tool is retracted by 1 mm.

Contour subprogram

• The contour will be entered as sequence of the commands G1, G2 and G3 in the contour subprogram.
  Programming chamfers and radii is allowed.
• The contour subprogram must contain at least 3 blocks with movements in both axes.
• The start point of the contour is the first position programmed in the contour subprogram.
• The commands G17, G18, G19, G41 and G42 and also frames are not allows in the subprogram.
• While roughing only the movements contained in the subprogram will be executed (only the contour will be machined).
• While finishing also the miscellaneous functions contained in the subprogram will be executed.

For face turning at the inner contour you have to select "**facing-outside**"!
The control regards "**facing-inside**" as a cycle that machines radially in +X -direction and axially in +Z-direction at the **rear** (clamped) face.
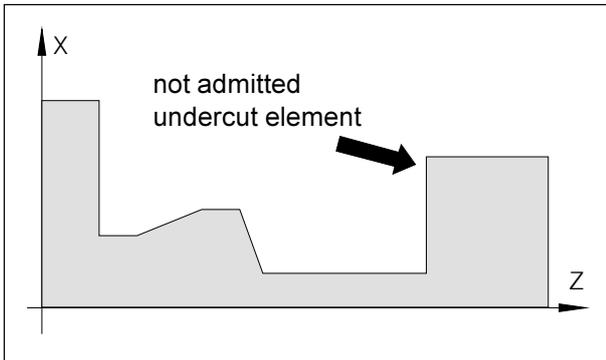
Contour monitoring

Following items will be monitored:

• Not admitted undercut elements.
  Undercut elements parallel to an axis are not admitted.
  Such contours can be machined with the grooving cycle.

• Clearance angle of the tool.
  When a clearance angle is entered in the tool data, it will be monitored, whether machining is possible with the active tool.
  When machining would result in a contour violation, machining will be aborted.
  When the clearance angle is entered in the tool data with the value 0, no monitoring occurs.

• Circle programming of arcs with a spread angle > 180°.
  Too large arcs also cause aborting the machining.
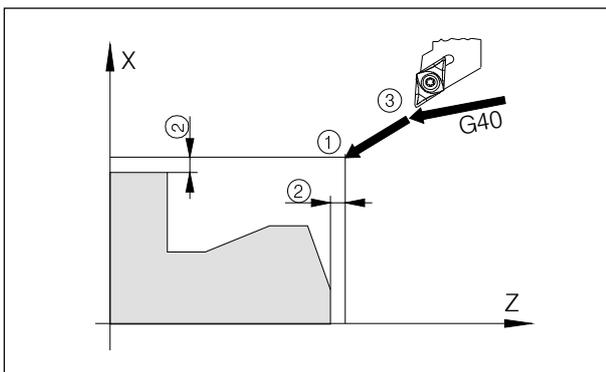
Start point

• The start point for machining (1) will be determined automatically.
  It is located outside the outest contour elements for {finishing allowance + 1 mm} (2).

• The tool position before cycle call (3) must be approached with G40 and must be located outside the rectangle that is spread by the first and the last point of the contour.

EMCO

Example CYCLE 95 longitudinal turning outside
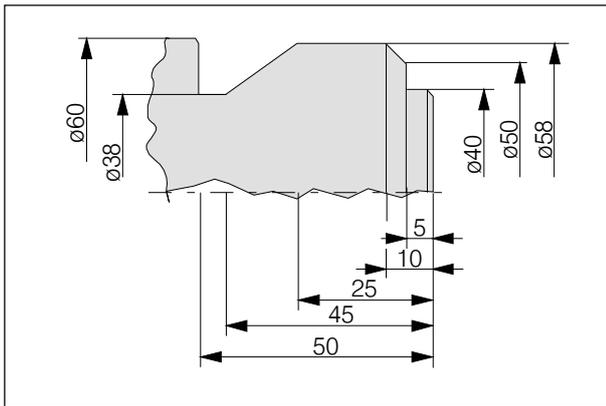
Name of the contour subprogram	CONT1
Infeed depth, without sign in radius	3
Finishing allowance longitudinal	0,05
Finishing allowance face in radius	0,3
Finishing allowance parallel to contour	0
Feed rate for roughing without undercut	0,3
Feed rate for dive-in in undercuts	0,1
Feed rate for finishing	0,12
Machining variant	9
Dwell time for chip breaking while roughing	0
Traverse path for roughing interruption, chip-breaking	0
Set-up clearance from the contour	0

Program:

G54	Zero offset
G53 G0 X610 Z350	Approach tool change position (without ZO)
T1 D1 G96 S250 M4	Tool call, cutting speed
G0 X65 Z0	Approaching to the workpiece
G1 F0,18 X-1,6	Face turning
G0 X65 Z5	Tool position before cycle
CYCLE95("CONT1",3,0.05,0.3,0,0.3,0.1,0.12,9,0,0,0)	Cycle call
G0 X200 Z100	Lift off
M30	Program end

Contour subprogram: CONT1:

G1 X38 Z2	Start point
Z0	First point at the contour (beginning chamfer)
X40 Z-1	
Z-5	
X50	
X58 Z-10	
Z-25	Contour points
X38 Z-45	
Z-50	
X60 CHR=0,3	
Z-50,4	
M17	Subprogram end

Example CYCLE 95 face turning outside
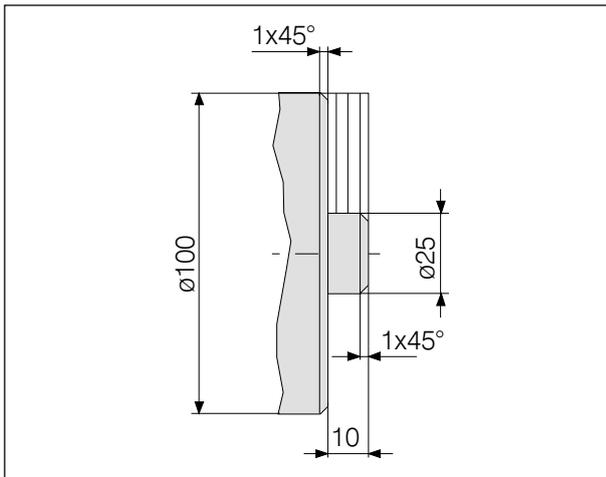
Name of the contour subprogram	CONT2
Infeed depth, without sign in radius	1
Finishing allowance longitudinal	0,02
Finishing allowance face in radius	0,05
Finishing allowance parallel to contour	0
Feed rate for roughing without undercut	0,3
Feed rate for dive-in in undercuts	0,1
Feed rate for finishing	0,12
Machining variant	10
Dwell time for chip breaking while roughing	0
Traverse path for roughing interruption, chip-breaking	0
Set-up clearance from the contour	0

Program:

G54	Zero offset
G53 G0 X610 Z350	Approach tool change position (without ZO)
T1 D1 G96 S250 M4 ; roughing tool	Tool call, cutting speed
G0 X65 Z0	Approaching to the workpiece
G1 F0,18 X-1,6	Face turning
G0 X65 Z5	Tool position before cycle
CYCLE95("CONT2",1,0.02,0.05,0,0.3,0.1,0.12,**10**,0,0,0)	Cycle call
G0 X200 Z100	Lift off
M30	Program end

Contour subprogram: CONT2

G1 X100 Z-12	Start point = first point at the contour
Z-10 CHR=1	Chamfer
X25	
Z0 CHR=1	Contour points
X22	
M17	Subprogram end

Note: This contour is programmed from the left to the right.

Example CYCLE 95 Longitudinal turning inside
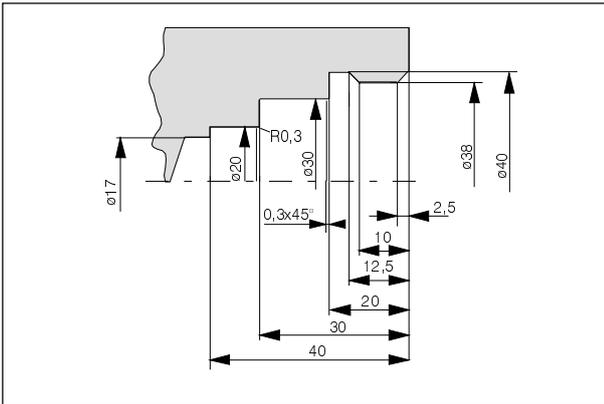
Name of the contour subprogram	CONT3
Infeed depth, without sign in radius	3
Finishing allowance longitudinal	0,05
Finishing allowance face in radius	0,3
Finishing allowance parallel to contour	0
Feed rate for roughing without undercut	0,3
Feed rate for dive-in in undercuts	0,1
Feed rate for finishing	0,12
Machining variant	11
Dwell time for chip breaking while roughing	0
Traverse path for roughing interruption, chip-breaking	0
Set-up clearance from the contour	0

Program:

G54	Zero offset
G53 G0 X610 Z350	Approach tool change position (without ZO)
T5 D1 G96 S250 M4 ; boring bar	Tool call, cutting speed
CYCLE95("CONT3",3,0.05,0.3,0,0.3,0.1,0.12,**11**,0,0,0)	Cycle call
G0 X200 Z100	Lift off
M30	Program end

Contour subprogram: CONT3

G1 X40 Z0 F0,12	Start point = first point at the contour
X38 Z-2,5	
Z-10	
X40 Z-12,5	
Z-20	
X30 CHR=0,3	Contour points
Z-30 F0,1	
X20 RND=0,3	
Z-40	
X17	
M17	Subprogram end

Example CYCLE 95 face turning inside
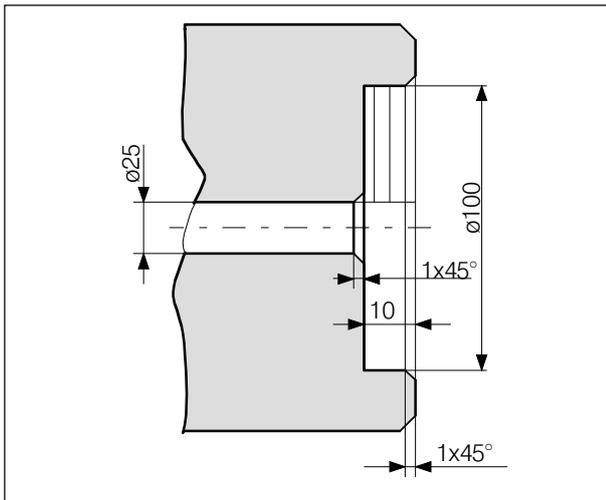
Name of the contour subprogram	CONT4
Infeed depth, without sign in radius	1
Finishing allowance longitudinal	0,02
Finishing allowance face in radius	0,05
Finishing allowance parallel to contour	0
Feed rate for roughing without undercut	0,3
Feed rate for dive-in in undercuts	0,1
Feed rate for finishing	0,12
Machining variant	10
Dwell time for chip breaking while roughing	0
Traverse path for roughing interruption, chip-breaking	0
Set-up clearance from the contour	0

Programm:

G54	Zero offset
G53 G0 X610 Z350	Approach tool change position (without ZO)
....	
T1 D1 G96 S250 M4 ; boring bar	Tool call, cutting speed
G0 X65 Z0	Approaching the workpiece4
CYCLE95("CONT4",1,0.02,0.05,0,0.3,0.1,0.12,**10**,0,0,0)	Cycle call
G0 X200 Z100	Lift off
M30	Program end

im Unterprogramm: CONT4

G1 X25 Z-12	Start point = first point at the contour
Z-10 CHR=1	
X100	Contour points
Z0 CHR=1	
X103	
M17	Subprogram end

Note: This contour is programmed from the left to the right.

EMCO

CYCLE 96 Thread undercut cycle

CYCLE96 (DIATH,SPL,FORM,VARI)

DIATH   nominal diameter of thread   **DIA**meter **TH**read
SPL   start point in Z   **S**tart **P**oint **L**ength
FORM   form of thread undercut   **FORM**
             Values:
             A-D: for Form A-D according DIN 76
VARI(*) Determination of the undercut position **VARI**ante
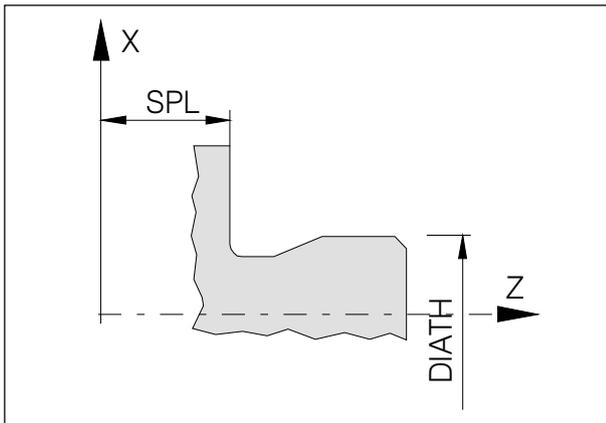
This cycle produces thread undercuts according DIN 76 of the form A - D for parts with metrical ISO threads in the size M3 to M68.
Undercuts (form E and F DIN 509) see CYCLE 94.

DIATH, SPL
DIATH indicates the nominal diameter of the thread. Thread undercuts below M3 and above M68 can not be produces with this cycle.
SPL indicates the final dimension (shoulder) in Z.
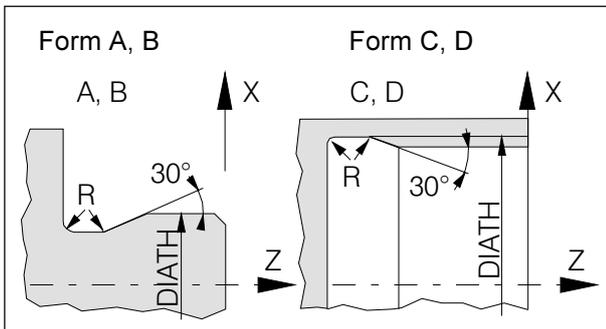
FORM
Form defines the kind of thread undercut according DIN 76.
Form A: for external threads
Form B: for external threads, short version
Form C: for internal threads
Form D: for internal threads, short version
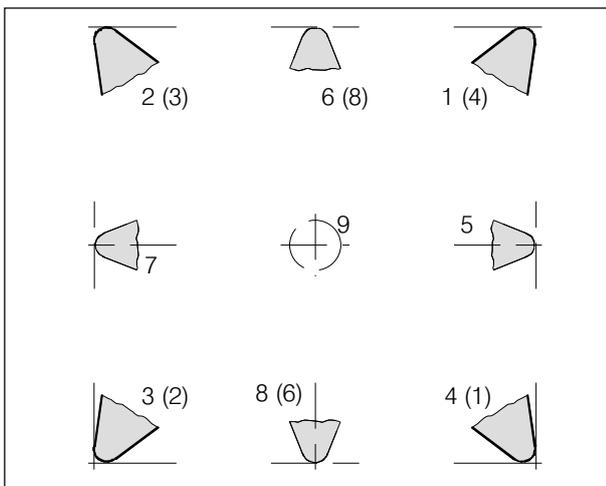
VARI
Only tools with the cutter positions 1, 2, 3, 4 can be used for this cycle.

When a clearance angle is entered in the tool data, it will be monitored.
After detecting that the form of the undercut can not be produced with the selected toolbecause of a too small clearance angle, the message: "changed form of undercut" will appear at the screen.
Machining will be continued (the error in form normally is very small).

For machines with the tool below (in front of) the turning axis (e.g. PC TURN 50/55), the values in brackets are valid.

EMCO

CYCLE 97 Thread cutting cycle

CYCLE97 (PIT,MPIT,SPL,FPL,DM1,DM2,APP,ROP,TDEP,FAL,IANG,
NSP,NRC,NID,VARI,NUMT,VRT)

PIT	thread pitch as value	**PIT**ch
MPIT	thread pitch as nominal size	**M**etrical **PIT**ch
	Thread pitch of regular metric thread, value 3 (M3) - 60 (M60).	

Program either MPIT or PIT. Contradictious values trigger an alarm.

SPL	start point of the thread in Z	**S**tart **P**oint **L**ength
FPL	end point of the thread in Z	**F**inal **P**oint **L**ength
DM1	diameter of the thread at the start point	
DM2	diameter of the thread at the end point	
APP	approach path without sign	**AP**proach **P**ath
ROP	run-out path without sign	**R**un **O**ut **P**ath
TDEP	thread depth without sign	**T**hread **DEP**th
FAL	finishing allowance without sign	**F**inishing **AL**lowance
IANG	infeed angle	**I**nfeed **ANG**le
	positive value: flank infeed at one flank	
	negativevalue: alternating flank infeed	
NSP	start point offset for the first thread without sign	
NRC	number of roughing cuts	**N**umber **R**oughing **C**uts
NID	number of idle cuts	**N**umber **ID**le cuts
VARI	machining variant	**Vari**ant
NUMT	number of threads	**NUM**ber **T**hreads
VRT⁽*⁾	Variable return distance from the contour	

Function:
- The thread cutting cycle produces straight or tapered external or internal threads with constant pitch.

- The threads can be single-threaded or multiple-threaded. Multiple-threaded threads will be produced one-by-one thread.

- Right-hand-thread or left-hand-thread is determined by the direction of rotation before cycle start.

- You can select either constant infeed per cut or constant cross-section of cut.

EMCO

Machining sequence:

• Approaching the start point at the begin of the approach path with G0.
• Infeed for roughing corresponding to VARI.
• Repeat roughing corresponding to NRC (number of roughing cuts).
• The following cut removes the finishing allowance with G33.
• Finishing will be repeated corresponding to NID (number of idle cuts).
• For every further thread the sequence will be repeated.
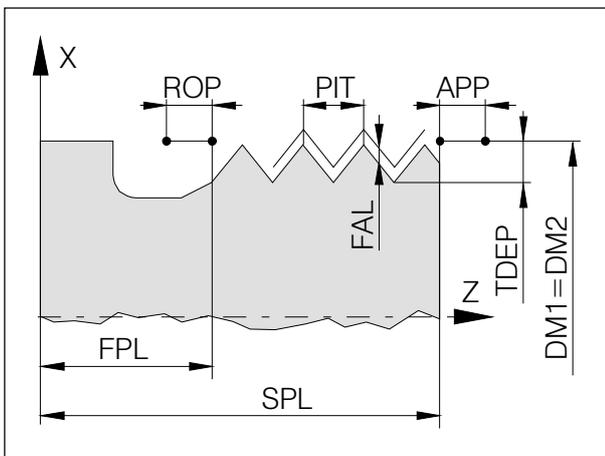
PIT, MPIT
The thread pitch is an axis-parallel value and will be entered without sign.
PIT defines the thread pitch in mm, MPIT as nominal value (M3 - M60) for regular metric threads.
Program either MPIT or PIT. Contradictious values trigger an alarm.

SPL, FPL, APP, ROP
The parameter SPL and FPL define the start and end point of the thread.
Machining the thread starts for APP (approach path) before SPL and ends for ROP (run-out path) after the thread.
Approach and run-out path are necessary to accelerate and slow down the slides.
In the approach and run-out area the thread is not precise, therefore thread undercuts should be used.
The start point in X for machining is 1 mm over the programmed thread diameter.

TDEP, FAL, NRC, NID
The finishing allowance FAL will be subtracted from the thread depth TDEP and the remaining rest will be divided in roughing cuts (number NRC). The division of the roughing cuts occurs according to VARI (constant or degressive).
Afterwards the finishing allowance FAL will be removed in one cut.
Subsequent occurs the number NID of idle cuts.

Note:
For regular metric threads:
Thread depth = 0,613435 x thread pitch
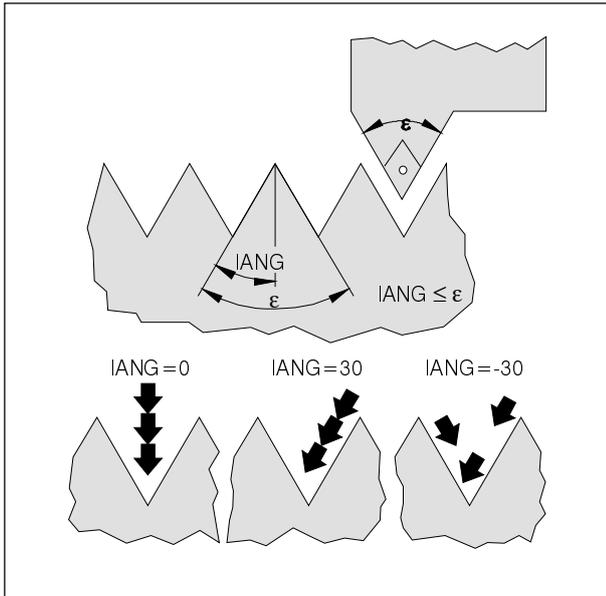
IANG
Infeed angle

Straight infeed
For straight infeed (vertical to the thread), program IANG = 0.

Flank infeed
The value IANG must be max. half the thread angle (e.g. for metric threads max. 30°).

Alternating flank infeed
A negative value for IANG causes alternating flank infeed.
With tapered threads a alternating flank infeed is not possible.

NSP
This angle determines the cut-in point of the first thread at the circumference of the workpiece.
If NSP is not programmed the thread starts at the 0°-position.
Input range 0.0001° to +359.9999°

VARI	O / I	Infeed
1	outside	constant infeed depth, taking down of the chip cross-section
2	inside	constant infeed depth, taking down of the chip cross-section
3	outside	constant cross- section of cut, taking down of the infeed depth
4	inside	constant cross- section of cut, taking down of the infeed depth

VARI
VARI determines outside / inside machining and the way of infeed.
VARI can have the values 1 to 4.

With division of the total infeed in single infeeds with constant chip cross-section (VARI 3, 4) the cutting pressure is constant for all roughing cuts. The infeed occurs with different values for each infeed depth.
For infeed with constant infeed depth (VARI 1, 2) the chip cross section increases from cut to cut.

EMCO
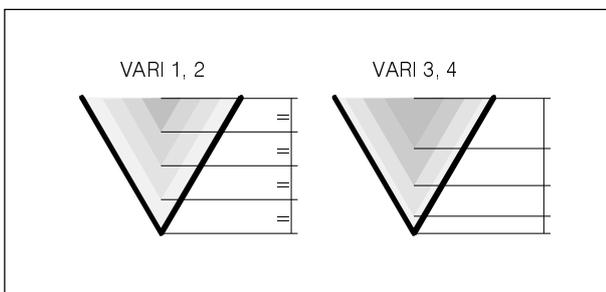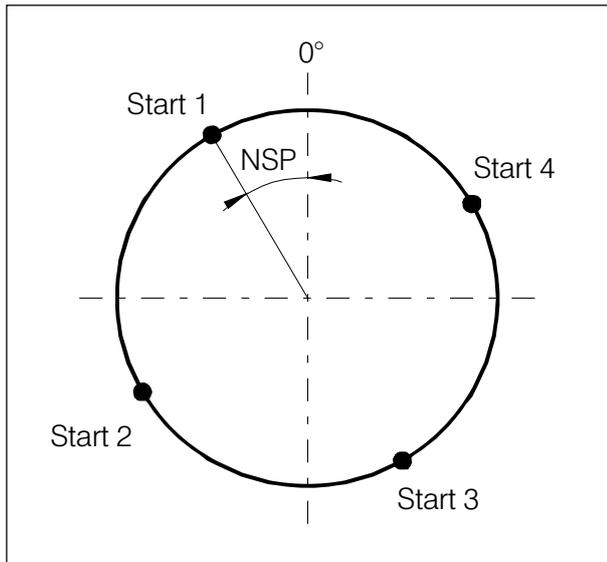
NUMT
Number of threads for multiple-threaded threads.

For a normal thread program 0 or do not program the parameter.

The single threads will be placed evenly on the circumference, the beginning of the first thread is determined by NSP.

To produce a multiple-threaded thread with irregular arrangement of the single threads you must program a separate cycle for every thread with a separate start position NSP.

VRT
Return path during threading. When VRT=0 (parameter not programmed) the tool is retracted by 1 mm.

Distinction longitudinal - face thread

If the taper angle of a tapered thread is ≤ 45°, the thread will be machined on the longitudinal axis, with taper angles over 45° the thread will be machined on the cross axis.
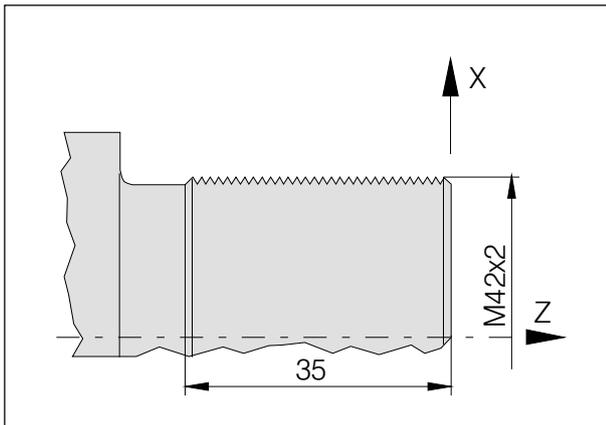
Example CYCLE 97 External thread

This program produces a metrical thread M42x4,5. The infeed is at the flank with constant chip cross-section.
5 roughing cuts will be executed with a thread depth of 2.76 mm without finishing allowance.
Afterwards 2 idle cuts will be done.

Thread pitch nominal thread size MPIT	M42
Start point longitudinal SPL	0
End point longitudinal FPL	-35
Thread diameter at the start point DM1	42
Thread diameter at the end point DM2	42
Appproach path APP	10
Run-out path ROP	3
Thread depth TDEP	2.76
Finishing allowance FAL	0
Infeed angle IANG	30
Start point offset NSP	0
Number of roughing cuts NRC	5
Number of idle cuts NID	2
Machining variant VARI	3
Number of threads NUMT	1
Variable return path VRT	1

Program:

G54	Zero offset
G53 G0 X610 Z350	Approach tool change position (without ZO)
T5 D1 G95 S1000 M4 ; thread tool	Tool call
G0 X44 Z12	Approach to workpiece
CYCLE97(,42,0,-35,42,42,10,3,2.76, ,30, ,5,2,3,1,1)	Cycle call
G0 X200 Z100	Lift off
M30	Program end

emco

CYCLE 98 Chaining of threads

CYCLE98   (PO1,DM1,PO2,DM2,PO3,DM3,PO4,DM4,APP,ROP,TDEP,FAL,IANG,
           NSP,NRC,NID,PP1,PP2,PP3,VARI,NUMT,VRT)

PO1	start point of the thread in Z	
DM1	diameter of the thread at the start point	
PO2	1st intermediate point of the thread in Z	
DM2	diameter of the thread at the 1st intermediate point	
PO3	2nd intermediate point of the thread in Z	
DM3	diameter of the thread at the 2nd intermediate point	
PO4	end point of the thread in Z	
DM4	diameter of the thread at the end point	
APP	approach path without sign	**AP**proach **P**ath
ROP	run-out path without sign	**R**un **O**ut **P**ath
TDEP	thread depth without sign	**T**hread **DEP**th
FAL	finishing allowance without sign	**F**inishing **AL**lowance
IANG	infeed angle	**I**nfeed **ANG**le
	positive value: flank infeed at one flank	
	negative value: alternating flank infeed	
NSP	start point offset for the first thread without sign	
NRC	number of roughing cuts	**N**umber **R**oughing **C**uts
NID	number of idle cuts	**N**umber **ID**le cuts
PP1	thread pitch 1 as value	
PP2	thread pitch 2 as value	
PP3	thread pitch 3 as value	
VARI	machining variant	**Vari**ant
NUMT	number of threads	**NUM**ber **T**hreads
VRT[(*)]	Variable return path from the contour	
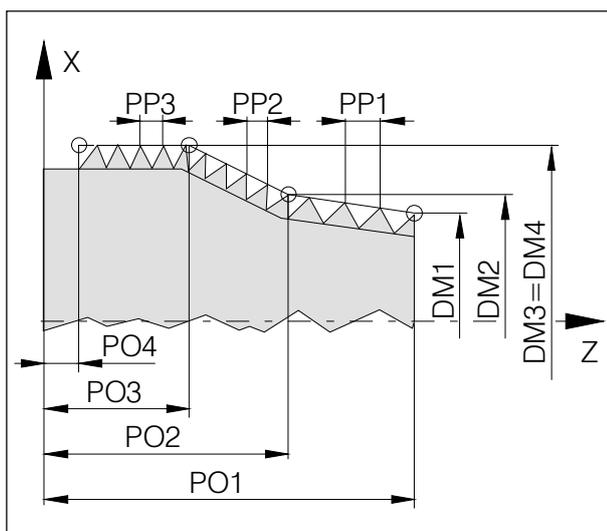
PO1, DM1 .. PO4, DM4, PP1, PP2, PP3
The parameter PO1, DM1 .. PO4, DM4 define the contour points of the thread chain.
The parameter PP1, PP2 and PP3 the pitches of the single thread sections.
All other parameter are the same as with the threading cycle CYCLE97.

The pitch between two tapered threads must not be exactly 45°. It always has to be <45°(less than) or >45°(greater than).

EMCO

Example CYCLE 98 Chaining of threads

This program producese a chain of threads, starting with a cylindrical thread.
The infeed is vertical to the thread with constant chip cross-section.
5 roughing cuts and 1 idle cut will be executed.
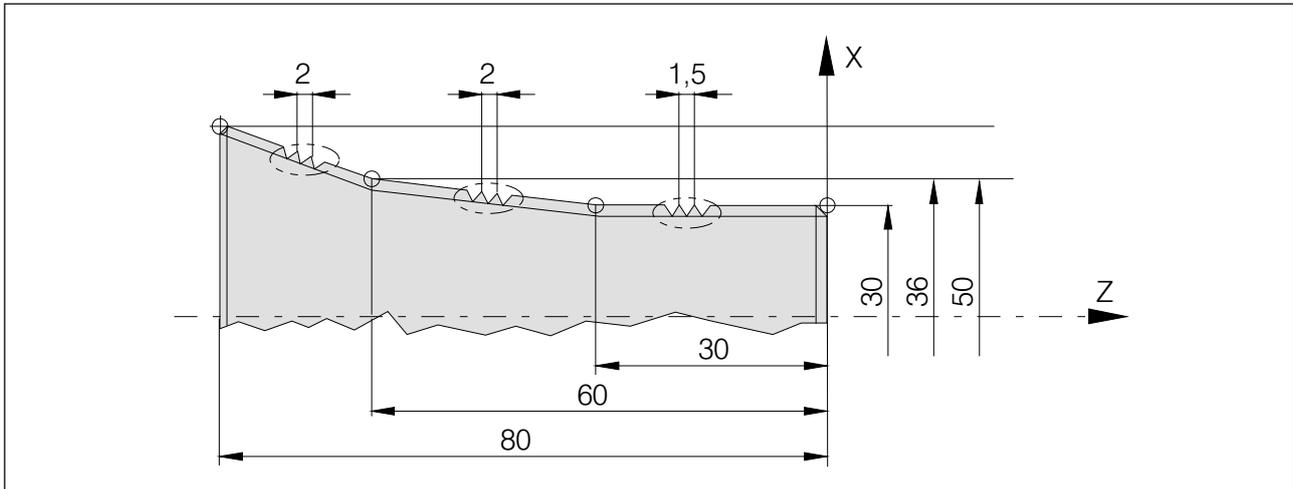
Start point longitudinal PO1	0
Diameter at the start point DM1	30
1st intermediate point PO2	-30
Diameter at the 1st intermediate point DM2	30
2nd intermediate point PO3	-60
Diameter at the 2nd intermediate point DM3	36
End point PO4	-80
Diameter at the end point DM4	50
Approach path APP	10
Run-out path ROP	10
Thread depth TDEP	0,92
Finishing allowance FAL	0
Infeed angle IANG	0
Start point offset NSP	0
Number of roughing cuts NRC	5
Number of idle cuts NID	1
Thread pitch 1	1,5
Thread pitch 2	2
Thread pitch 3	2
Machining variant VARI	3
Number of threads NUMT	1
Variable return path VRT	1

Program:

G54	Zero offset
G53 G0 X610 Z350	Approach tool change position (without ZO)
T5 D1 G95 S1000 M4 ; thread tool	Tool call
G0 X32 Z12	Approach to workpiece
CYCLE98(0,30,-30,30,-60,36,-80,50,10,10,0.92, , , ,5,1,1.5,2,2,3,1,1)	
	Cycle call
G0 X200 Z100	Lift off
M30	Program end

EMCO
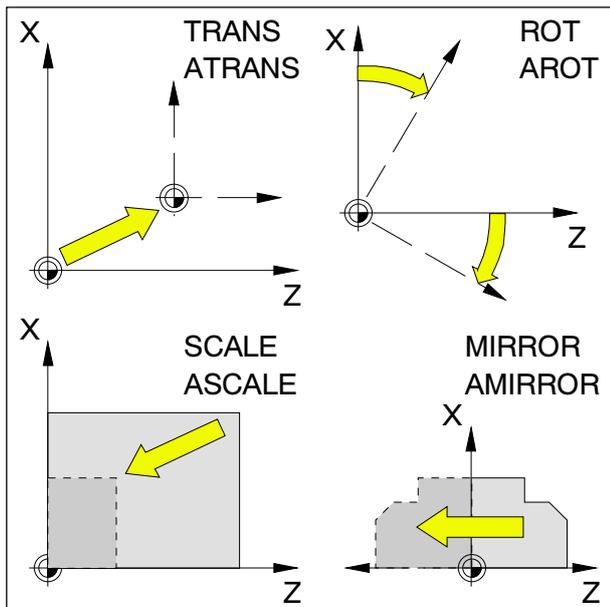
Frames

Frames alter the actual coordinate system.

- Shift coordinate system: TRANS, ATRANS
- Rotate coordinate system: ROT, AROT
- Programmable scale factor: SCALE, ASCALE
- Mirror coordinate system: MIRROR, AMIRROR

The frame commands will be programmed in a separate NC block and executed in the programmed sequence.
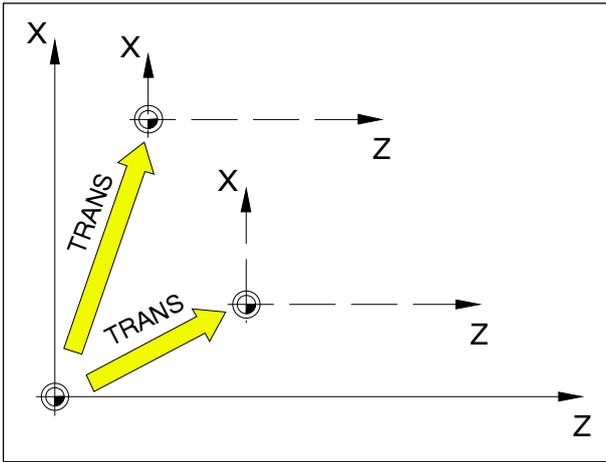
ATRANS relates to the last valid zero point G54 - G599, TRANS.

Programmable zero offset TRANS, ATRANS

Format:
TRANS/ATRANS   X...   Z...

TRANS   Zero offset absolute, related to the actual zero point G54-G599.
(TRANS deletes all previous programmed frames (TRANS, ATRANS, ROT, AROT, ...)).

ATRANS   Zero offset additive, related to the actual settable (G54-G599) or programmed (TRANS/ATRANS) zero point.
A zero shift that builds-up on existing frames (TRANS, ATRANS, ROT, AROT, ...) is programmed with ATRANS.
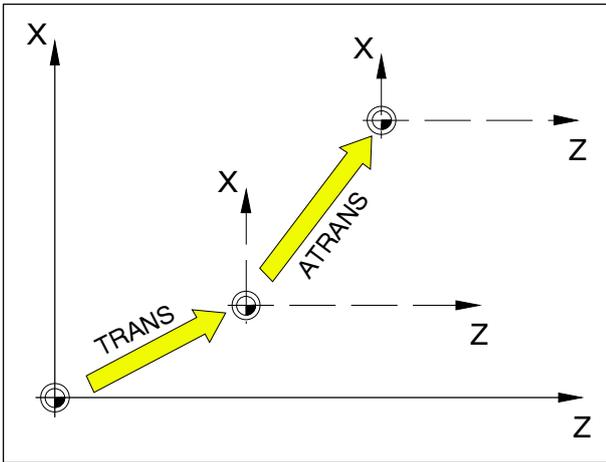
TRANS relates always to the actual zero point G54 - G599.
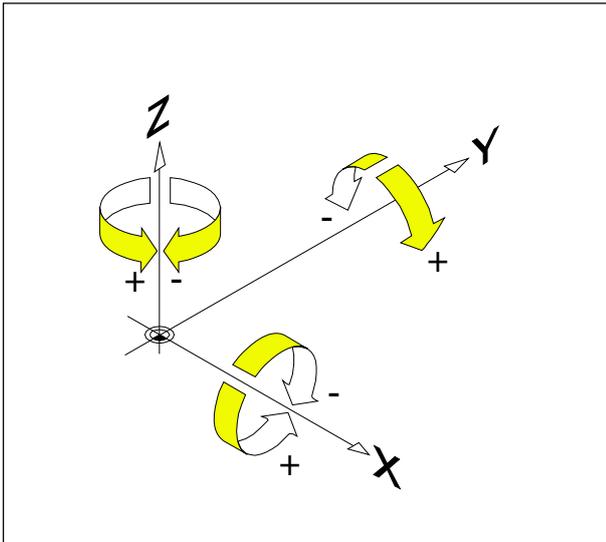
Programmable rotation ROT, AROT

ROT/AROT is used to rotate the workpiece coordinate system around each of the geometry axes X, Z or through an angle RPL in the selected working plane G18.

This allows easier programming of contours with main axes that are inclined to the geometry axes.

Format:

ROT/AROT   X..   Z..

ROT/AROT   RPL=..

ROT	Rotation absolute, related to the actual zero offset G54-G599. (ROT deletes all previous programmed frames (TRANS, ATRANS, ROT, AROT, ...)).
AROT	Rotation additive, related to the actual settable (G54-G599) or programmed (TRANS/ATRANS) zero offset. A rotation that builds-up on existing frames (TRANS, ATRANS, ROT, AROT, ...) is programmed with AROT.
X, Z	Rotation in space (in degrees); geometry axis around which the rotation takes place.
RPL=	Rotation in the plane (e.g. G17) (in degrees).

emco
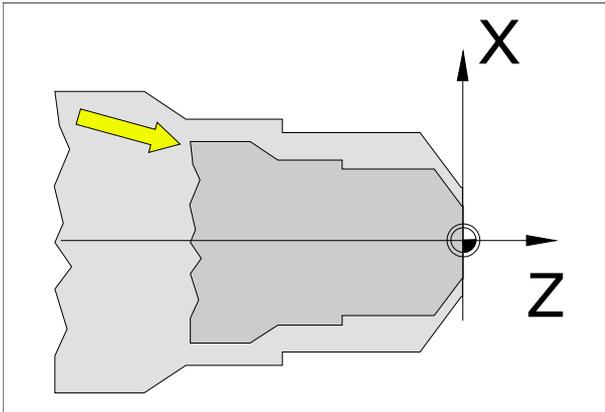
Programmable scale factor SCALE, ASCALE

SCALE/ASCALE allows to set a separate scale factor for every axis X, Z.
When different scale factors are used for X, Z the contour becomes distorted.

Format:

SCALE/ASCALE   X..   Z..

When after SCALE/ASCALE a zero offset is programmed with ATRANS it also will be scaled.

SCALE	Scale absolute, related to the actual settable zero offset G54-G599. SCALE deletes all previous programmed frames (TRANS, ATRANS, ROT, AROT, ...). SCALE without axis address deselects the scale factor (and all other frames).
ASCALE	Scale additive, related to the actual settable (G54-G599) or programmed (TRANS/ATRANS) zero point. A scale that builds-up on existing frames (TRANS, ATRANS, ROT, AROT, ...) is programmed with ASCALE.
X, Z	Scale factor for each axis.
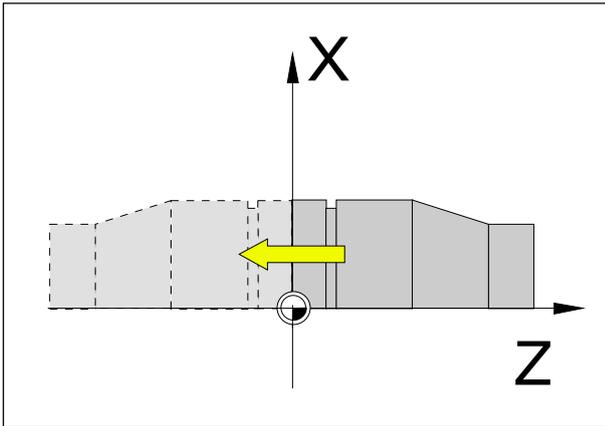
Programmable mirroring, MIRROR, AMIRROR

MIRROR/AMIRROR mirrors workpiece shapes on coordinate axes X, Z.

Format:

MIRROR/AMIRROR X..   Z..

When a contour is mirrored, the circle direction G2/G3 and the cutter radius compensation G41/G42 are changed automatically.

MIRROR	Mirroring absolute, related to the actual settable zero offset G54-G599. (MIRROR deletes all previous programmed frames (TRANS, ATRANS, ROT, AROT, ...)). MIRROR without axis address deselects mirroring (and all other frames).
AMIRROR	Mirroring additive, related to the actual settable (G54-G599) or programmed (TRANS/ATRANS) zero point. Mirroring, that builds-up on existing frames (TRANS, ATRANS, ROT, AROT, ...) is programmed with AMIRROR.
X, Z	Geometry axis to be mirrored on. The value indicates the distance from the mirror axis to the geometry axis, e.g. X0.

EMCO
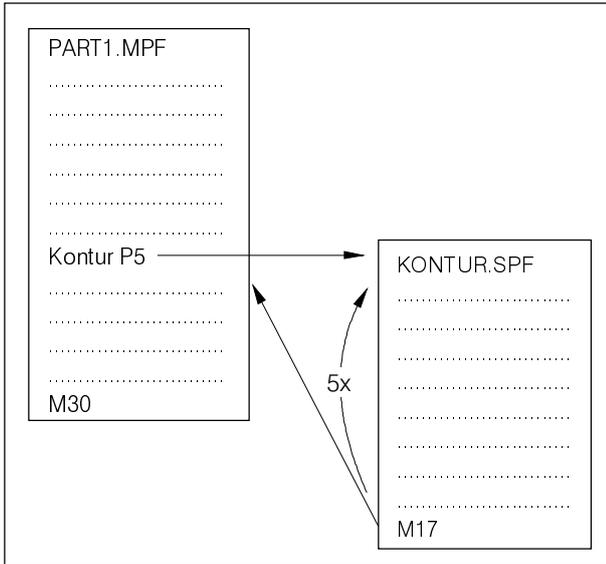
Subprograms

Program run with subprogram

Functions which are repeated multiple can be programmed as subprograms.

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

R parameter can be transvered in subprograms

Subprogram Call in Part Program

e.g.:   Mill1 P1 LF
Mill1   Subprogram
P1   Number of Subprogram runs
           (max. 99)

Subprogram End with M17

e.g.:   N150 M17 LF
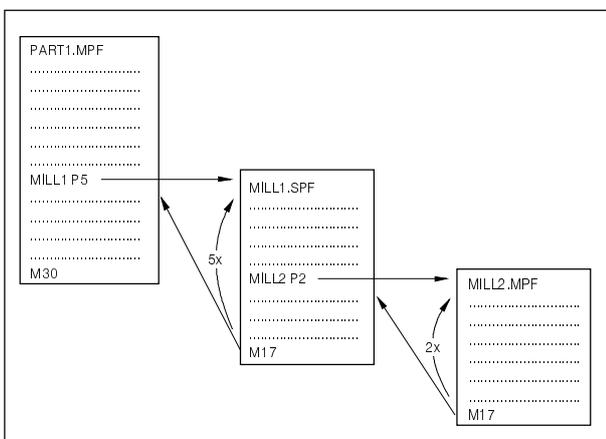
Subprogram nesting
A eleven-fold nesting of subprograms is possible. Block search is possible into the eleventh subroutine level.

Cycles also count as subprograms, that means e.g. a drilling cycle can be called max. in the 10th subprogram level.

Nesting of subprograms

Subprogram with SAVE- mechanism

With this function, the operating data which are currently valid in the main program, such as G functions or overall Frame, are stored when the subprogram is called. On return to the calling program the old state is automatically restored.
For this, specify the additional command SAVE with the definition statement with PROC.

Subprograms with passing parameters

Beginning of program, PROC

A subprogram that is to take over parameters from the calling program when the program runs is designated with the vocabulary word PROC.

End of program M17, RET

The command M17 designates the end of subprogram and is also an instruction to return to the calling main program.
The vocabulary word RET stands for end of subprogram without interuption of continous path mode and without function output to the PLC.

Subprogram calls must be programed in a seperate NC block.
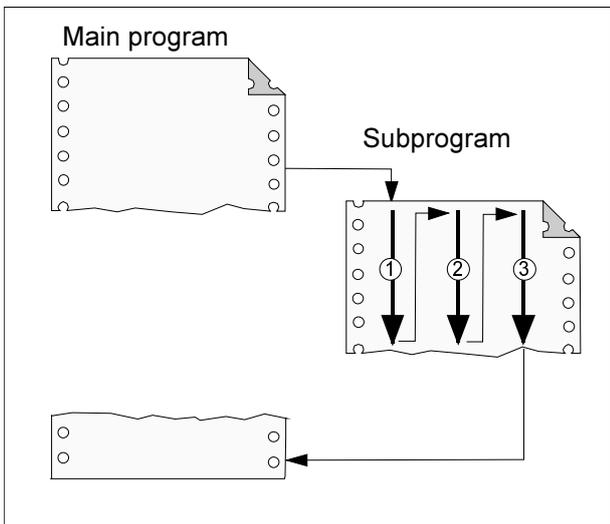
Subprogram with program repeating, P

If you want to execute a subprogram several times in succession, you can program the required number of program repetitions in the block in the subprogram call under address P.
Parameters are only passed on during the program call or the first pass. The parameters remain unchanged for the repetitions.

Modal subprogram MCALL

With this fnction the subprogram is automatically called an executed after every block with motion.
In this way you can automate the calling os subprograms that are to be executed at different positions on the workpiece. For exanble, for drilling patters.

IIn a program run, only one MCALL call can apply at any one time. Parameters are only passed once with MCALL.

Main program

N10 G0 X0 Yo
N20 MCALL L70
N30 X10 Y10

Subprogram L70
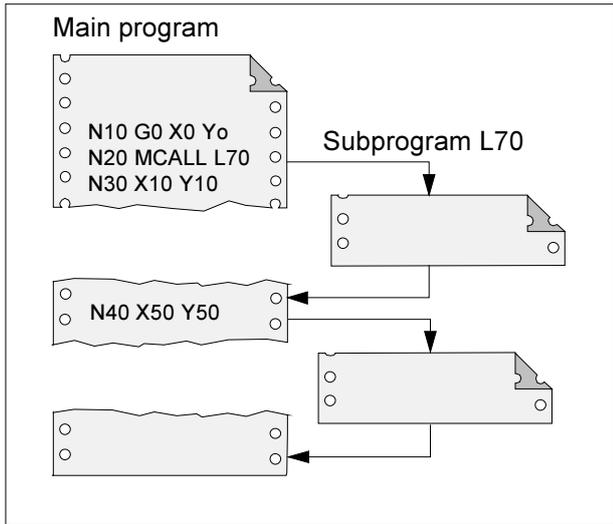
N40 X50 Y50

Example

N10 G0 X0 Y0
N20 MCALL L70
N30 X10 Y10
N40 X50 Y50

Deactivating the modal subprogram call

With MCALL without a subprogram call or by programming a new modal subprogram call for a new subprogram.

Program jumps

Uncontitional program jumps

Format

Label:
GOTOB LABEL
or
GOTOF LABEL
Label:

GOTOB   Jump instruction with jump destination backwards (towards the start of program)
GOTOF   Jump instruction with jump destination forwards (towards the end of program)
LABEL   Destination (label within the program)
LABEL: Jump destinaltion

Programs working in standard manner (main programs, subroutines, cycles,..) can be changed in order by means of program jumps. Destination addresses can be approached within a program by means of GOTOF and/or GOTOB.
The program continues processing with the instruction following immediately the destination address.

Program jumps must be programed in a seperate NC block.

Conditional program jumps

Format:

Label:
IF expression GOTOB LABEL
oder
IF expression GOTOF LABEL
LABEL:

IF   Conditions

GOTOB   Jump instruction with jump destination backwards (towards the start of program)
GOTOF   Jump instruction with jump destination forwards (towards the end of program)
LABEL   Destination (label within the program)
LABEL: Jump destination

Jump conditions can be formulated with IF statements. The jump to the programmed destination only occurs if the jump condition is fulfilled.

EMCO

Programming messages, MSG

Messages can be programmed to provide the user with information about the current machining situation during program execution.

A message is generated in an NC program by inserting the keyword "MSG" in parentheses "()" followed by the message text in double quotation marks.

A message can be cleared by programming "MSG()".

> A message text can be up to 124 characters long and is displayed in two lines (2x62 characters). Contents of variables can also be displayed in message text.

Example:
```
N10  MSG ("Roughing of contour")
N20  X... Y...
N ...
N90  MSG ()
```

You can also set alarms in addition to messages in an NC program. Alarms are displayed in a separate field on the screen display. An alarm is assocoated with a reaction on the control which depends on the alarm category.
Alarms are programmed by inserting the keyword "SETAL" followed by the alarm number in parentheses.
Alarms are always programmed in a seperate block.

Example:
```
N100  SETAL (65000)          ;Set alarm 65000
```

EMCO

C axis

For milling surfaces (square, hexagon etc.) the C axis and the tool slide must be moved against each other in a definite relation (=hobbing).

Such surfaces can be programmed easily with the software accessory "TMCON".

For description and programming examples see chapter "Programming/TMCON".

Switching on and positioning the C axis

SPOS[1]=0	switch on C- axis and positioning 0°
G0 C90	C- axis on 90°

Deselection of the C axis

M3, M4, M5

JOG operation of the C axes

To be able to operate the C axes in JOG operation, the following program must be carried out before in MDA operative mode:

Main spindle

SPOS=0	(switch on C axis and position to 0)
G0 C0	(C axis movement)
M30	

PC Turn 155
In the JOG mode it is not possible to work with the C- axis.

Positioning spindles SPOS, SPOSA

The programming of spindle position must be programed in a seperate NC block.

SPOS=... or SPOS [n]=
M70 or Mn=70
SPOSA=... or SPOSA [n]=
WAITS or WAITS (n,n,n)

SPOS/SPOS[n] .... Position master spindle or spindle with number n. NC block is not enable until the position has been reached.

M70/Mn=70 .... Switch over master spindle or spindle with number n to axis operator. No defined position is approached.

SPOSA/SPOSA[n] Positio master spindle or spindle number n. The next NC block is enabled, even if the position has not been reached.

WAITS/WAITS(n,n,n) Wait for spindle position to be reached. WAITS applies to the master spindle or the specified spindle number.

SPOS/M70 and SPOSA can be used to position spindles at specific angle locations, e.g. for tool change. The spindle can also be traversed as a path axis at the address specified in the machine data. The machine data for selected spindle are used immediately when M70 is programmed. When the axis name is specified, the spindle is in axis mode.
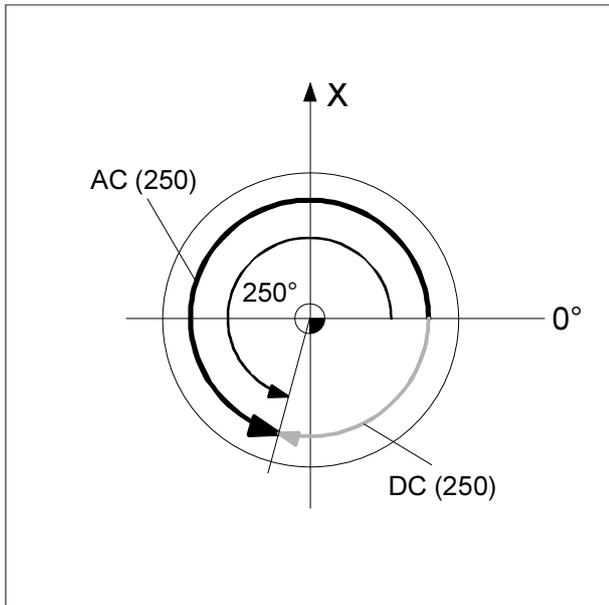
Specify spindle position:
The spindle position is specified in degrees. Since the commands G90/91 do not apply here, the following explicit references apply:

AC(...) .... Absolute dimension
IC(...) .... Incremental dimension
DC(...) .... Approach absolute value directly
ACN(...) ..... Absolute dimension, approach in negative dimension
ACP(...) ..... Absolute dimension, approach in positive direction.

Example.: N10 SPOSA [2] =ACN (250)

   Position spindle 2 at 250° in negative direction.

When no parameter is specified, traversing is automatic as with the DC parameter. Three spindle positions can be specified per NC block.

Note:
SPOS and SPOSA are effective until the next M3, M4, M5.
If instead of SPOS the spindle has been switched off with SPCON, it has to be switched on again with SPCOF.

Synchronize spindle movements:
WAITS, WAITS (n,n,n)

WAITS can be used to identify a point at which the NC program waits until one more spindles programmed with SPOSA in a previous NC block have reached their positions.

Bsp.:   N10 SPOSA [2] =180 SPOSA [3]=0
            N20...N30
            N40 WAITS (2,3)

   The block waits until spindles 2 and 3 have reached the positions specefied in block N10

Note:
When M3 or M4 are active, the spindle in the programmed value comes to a standstill.

If the spindle has not yet been synchronized with synchronization marks, the positive direction of rotation is taken from the machine data (state on supply)

EMCO

WAITP(...)

WAITP can be used for:

- Identifying a position in the NC program where the program is to wait until an axis programmed with POSA in a previous NC block has reached its end positions.
- Making an axis available as a reciprocating axis.
- Making an axis available for traversing as a concurrent positioning axis.

After WAITP, assignment of the axis to the NC program is no longer valid; this applies until the axis is programmed again.

emco

Extended addresses of Spindle speed S and Spindle rotation M3, M4, M5,SETMS

Spindle 1 = Masterspindle (= on-position)

Spindel 1
(main spindle)

Spindel 2
(tool spindle)
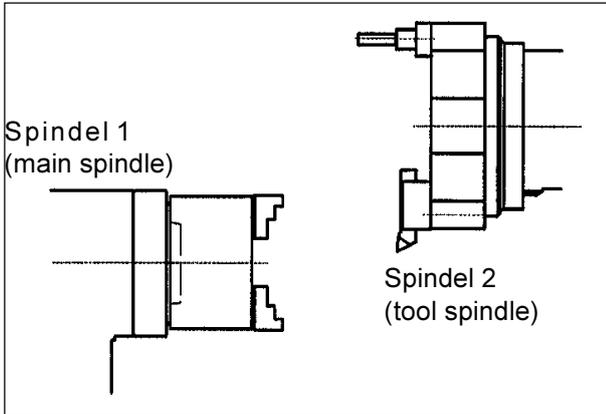
S...M3	main spindle right, speed S...
S...M4	main spindle left, speed S...
M5	main spindle Stop
S2=... M2=3	tool spindle right, speed S...
S2=... M2=4	tool spindle left, speed S...
M2=5	tool spindle Stop

Example 1

The main spindle remains master spindle:
The spindle number of the driven tool must be programmed additionally.

S2000 M3   Main spindle on
T1 D1   tool T1
G94 S2=1000 M2=3 speed for driven tool

        └── direction M3
       └── spindle number 2
   └── Only G94(mm/min) possible.
     With G95(mm/rev) the feed would relate to the speed of the master spindle (=main spindle)

Spindel 2 = Masterspindle

Spindel 1
(main spindle)

Spindel 2
(tool spindle)
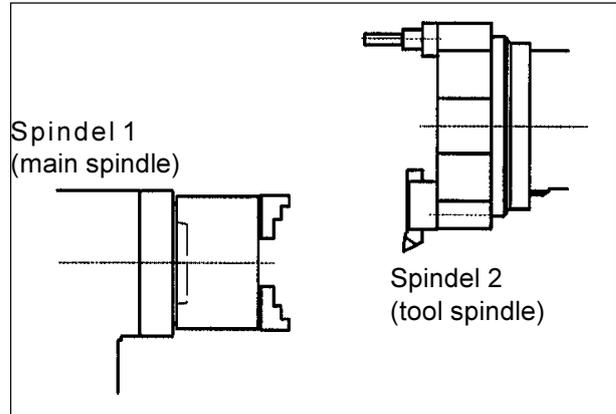
S1=... M1=3	main spindle right, speed S...
S1=... M1=4	main spindle left, speed S...
M1=5	main spindle Stop
S...M3	tool spindle right, speed S...
S...M4	tool spindle left, speed S...
M5	tool spindle Stop
SETMS(2)	Spindle 2 remains Masterspindle
SETMS	reset to on-position

Example 2

The tool spindle is defined as master spindle:
The driven tools are programmed like a main spindle.

T1 D1   tool T1 tool correction
SETMS(2)   spindle 2 is master spindle
SPOS[1]=0   activate C axis
G95 S1000 M3   speed for driven tool

 └── G95(mm/rev) or G94(mm/min) possible.
  G95 relates to the speed of the master spindle (=tool).
  Thread cutting with thread taps without length compensation is also possible.

EMCO

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

TRANSMIT

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

Any contour can be milled at the plane face of workpieces by means of Transmit.

Selection:
general .... TMCON

Deselection:
general .... TMCOFF

TMCON and TMCOFF are stored under the usercycles and free programmable.

Example- Transmit (Hexagon Key- size 30)

```
G54
TRANS Z100
TMCON
T3  D1                          (end-milling cutter DM 5-
                                tool type 100; L1=Z - L3=X)
G94  S1000  M3  F120
G0  X45  Y10
X17.32  Y10  G41
Z-6
G1  Y0
X8.66  Y-15
X-8.66
X-17.32  Y0
X-8.66  Y15
X8.66
X17.32  Y0
Y-10
G40
Z100 M5
TMCOFF                          (Delselection of
                                Transformation)
M30
```

Note:
Due to the programmed G17 (i the programm TMCON) during the tool measurement, the Z-value must be programmed for L1 and the X-value for L3.
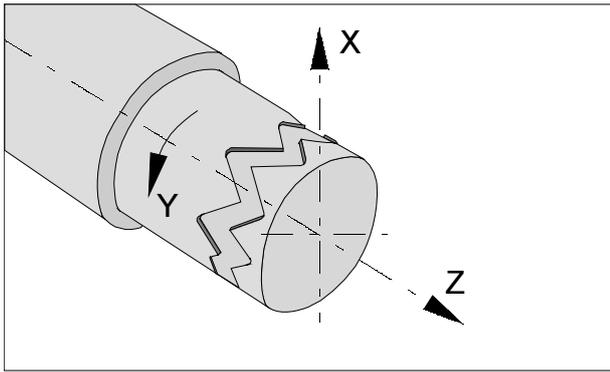
Note:
Due to an actual transformation or deselection of transformation, the zero point offset and the previous transformations (e.g. Transmit) are deselected and must be programmed again.

Gefräst mit Schaftfräser ø5mm
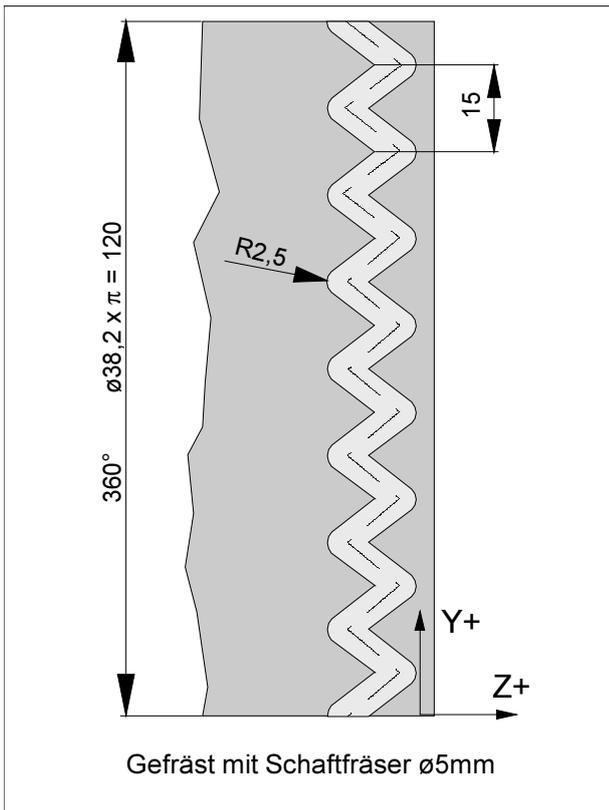
TRACYL

Is used for contour milling at the surface area.

The cylinder surface curve transformation provides the following capabilities:
   • Longitudinal grooves on cylindrical bodies,
   • Transverse grooves on cylindrical bodies,
   • Any other groove shapes on cylindrical bodies.

The shape of the grooves is programmed with reference to the processed level cylinder surface area.

Selection:
general ....TRACYL( )
Deselection:
general .... TRAFOOF

Example- Tracyl

```
G54
TRANS  Z150
T7  D1                      (end-milling cutter DM 5-
                            Tool type 100)
G19
SETMS (2)                   (Power tool)
G95  S1000  M3
G0  X45  Z0
SPOS [1] =0                 (Mainspindleat 0)
TRACYL (38.2)              (Workpiece- ø)
G54
TRANS  Z150
G1  X35  Y0  Z0  F0.3
G1  Z-10  Y7.5
Z0  Y15
Z-10  Y22.5
Z0  Y30
Z-10  Y37.5
Z0  Y45
Z-10  Y52.5
Z0  Y60
Z-10  Y67.5
Z0  Y75
Z-10  Y82.5
Z0  Y90
Z-10  Y97.5
Z0  Y105
Z-10  Y112.5
Z0  Y120
X45
TRAFOOF                     (Deselection of
                            transformation)
G54
TRANS  Z150
G0  X100  Z0
M30
```

Feed optimizing CFTCP, CFC, CFIN

Basic status (CFC):
With active cutter radius compensation G41/42 the feed is valid at the programmed contour.

The basic setting CFC can result in unwanted high or low feedrates in curves at the workpiece side opposite to the contour.
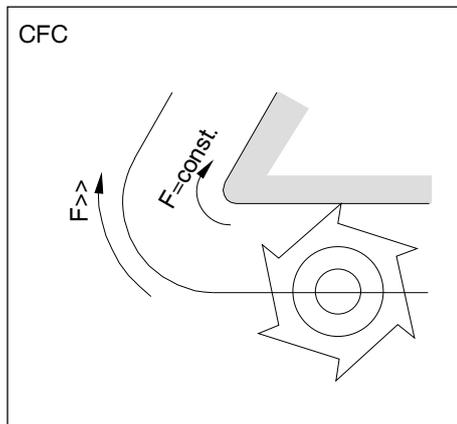
Tool outside the curve
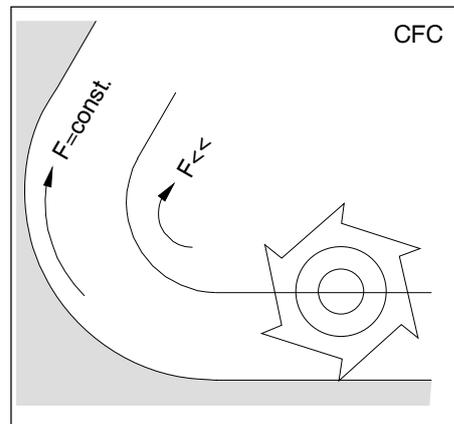
Tool inside the curve

The feed characteristic can be determined with the following commands

CFTCP

(**C**onstant **F**eed in **T**ool **C**entre **P**oint)
The shape of the contour does not influence the feed rate at the tool centre.
Application:
The tool cuts at the whole diameter.
(e.g. roughing)

CFC

(**C**onstant **F**eed at **C**ontour)
Basic setting.
Constant feed at the curve.
The feed rate of the tool centre path will be increased when the tool is outside a curve and reduced when the tool is inside a curve.
Application:
The tool cut oly at the circumference.
(e.g. finishing)

CFIN

(**C**onstant **F**eed at **IN**ternal radius)
The feed rate of the tool centre path will be reduced when the tool is inside a curve.
Outside curves do not increase the tool path feed rate (important for machining with fully invaded tool, end face finishing etc.).

Command description M-Commands

M00 Programmed Stop

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

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

M01 Programmed Stop, Conditional

M01 works like M00, but only when the function "PROGRAMMED STOP YES" was switched on via 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.

M02=3   **Driven tools On Clockwise**

M02=4   **Driven tools On Counterclockwise**

M02=5   **Driven tools Off**

M03 Main Spindle ON Clockwise

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

M04 Main Spindle ON Counterclockwise

The same conditions as described under M03 apply here.
M04 must be used for all left hand cutting tools or normal clamped tools if the tool is clamped behind the turning centre.

M05 Main Spindle OFF

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

M06 Tool change
M- Code for tool changing

M08 Coolant ON

Only for machines with coolant device.
The coolant will be switched on.

M09 Coolant OFF

Only for machines with coolant device.
The coolant will be switched off.

M10 Spindle brabe ON

Spindle brake get aktivate.

M11 Spindle brake OFF

Spindle brake get open.

M17 Subprogram End

M17 will be written in the last block of a subprogram.
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 automatic tailstock.
The tailstock sleeve traverses back.
See H: Accessory functions

M21 Tailstock FORWARD

The tailstock sleeve traverses forward.
See H: Accessory functions

M23 Colleting tray backward

M24 Collecting tray forward

M25 OPEN Clamping Device

The clamping device opens.
See H: Accessory functions

EMCO

⚠

Caution:
When M commands are programmed that can not be executed by the machine, the respective M command will be ignored and the program continues.
This can cause collisions (e.g. with missing workpiece manipulation).

M26 CLOSE Clamping Device

The clamping device closes.
See H: Accessory functions

M30 Main Program End

With M30 all drives will be switched off and the control will be resetted to program start.
Furthermore the workpiece counter will be increased by 1.

M71 Puff Blowing ON

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

M72 Puff Blowing OFF

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

EMCO

Free contour programming

The free contour programming is a support tool for the editor.

An integrated contour calculator calculates possibly missing parameters as soon as they result from other parameters.

Additionally, contour transition elements such as chamfer or radius are available.

The selection is carried out in the operation area Program .

You select an already existing proram and/or open a new part program via the softkeys "Workpiece" and "Part programs"

You open the contour editor with the softkeys "Support" and "New contour" and/or "Recompile" .
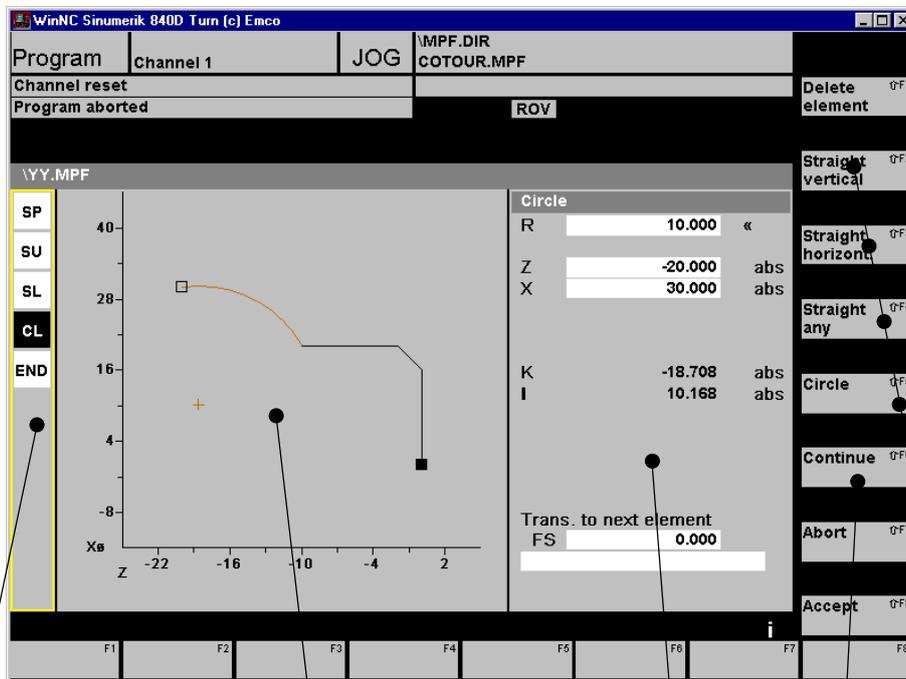
Programmed contour elements (can be opened again by double click)

Graphic representation of the programmed contour elements

Programming window

Contour elements

Further contour functions (close pole, contour)
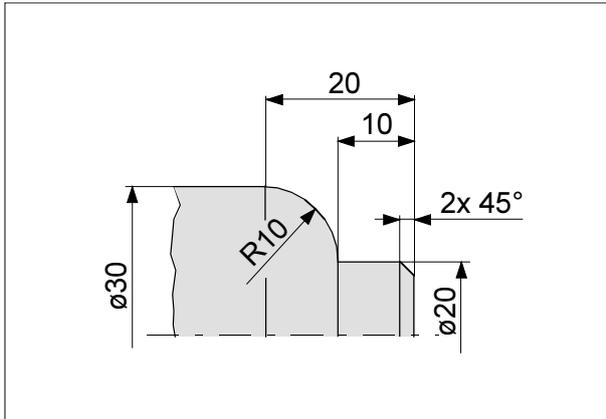
Programming example

Contour programming:

Determine starting point:

With the input of contours the starting point is determined at first.
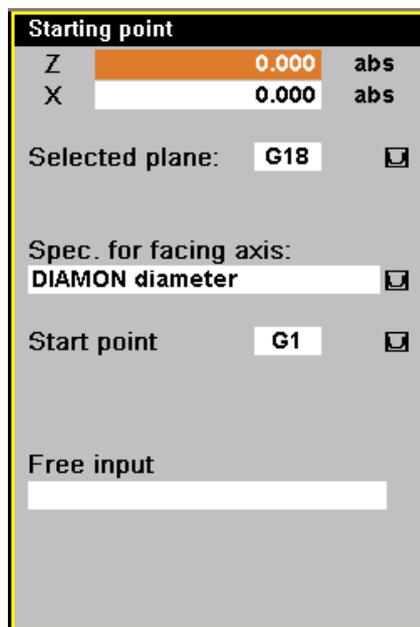
* The coordinates for X and Z must be programmed absolutely.

* Plane selection: G17 / G18 / G19
* Indication dimension facing axis
  - DIAMON (diameter)
  - DIAMOF (radius)
  - DIAM90 (diameter/radius)

* Start point (approach): G0 / G1

The programmed values are accepted in the editor with the softkey "accept element".

Further parameters for the contour definition are opened with the softkey "All parameters".

You can select between the selection options with the space bar or the softkey "Alternative".

Contour description "Straight vertical"
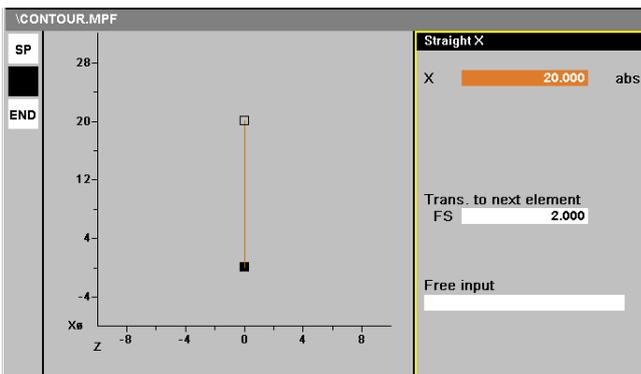
Selection softkey "Straight vertical"

End point X 20.000
Chamfer/radius/undercut FS 2.000

During the transition to the next element you have the possibility to select between a chamfer or a radius.

The contour description is accepted in the editor with the softkey "accept element".
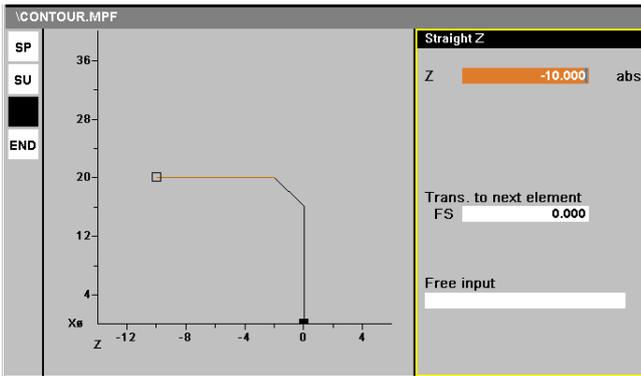
Contour description "Straight horizontal"

Selection softkey "Straight horizontal"

End point X .... -10.000
Transition to next element 0.000

The contour description is accepted in the editor with the softkey "accept element".
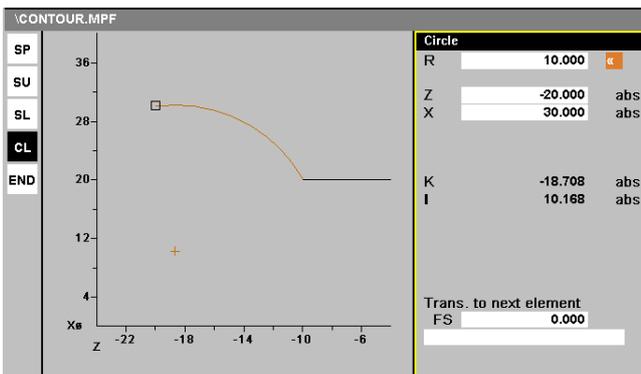
Contour description "Circle"

Selection softkey "Circle"

radius 10
End point Z .... -20.000
End point X 30
Transition to next element 0.000

With complete indication of radius and end point the values for I and K are calculated automatically.

The right selection of the propositions can be made with the softkey "Dialog select".   .
The selection is accepted with "Dialog accept".
The contour description is accepted in the editor with the softkey "Accept element".

You change from free contour programming back to the editor with the softkey "Accept".

During retranslation only the contour elements created with free contour programming are generated again. Texts carried out subsequently in the program text get lost during this procedure.

A contour already existing can be modified by means of the softkey "Recompile". During this procedure the cursor of the editor must be positioned within the contour.

EMCO

Softkey "Straight any"

Any straight lines can be programmed with this function.

Any straight line is an oblique line in X or Z- direction the end point of which is programmed via a coordinate point or an angle.

Unless parameter input fields are programmed, the control presumes that those values are unknown and tries to calculate them from other parameters.

Softkey "Continue"

You find the softkey "Pole" and "Close contour" under the softkey "Continue".

The softkey "Pole" is not active.

The softkey "Close contour" is used for closing a contour with <u>one</u> straight line.

Differences contour processor
EMCO – Siemens
KP Version 1.0.5

Graphic
- alternative solution is not indicated
- different scaling logic
- chamfers are only drawn between straight lines
- Undercuts are not drawn (are generally not implemented)

Inputs
- 'tangential' is represented as $a_2 = 0.000$
- Softkey 'Tangent prev. elem" is represented activated for elements with tangential transition
- 'Close contour' creates only one (any) and not two straight lines (vertical and horizontal)
- it is not possible to work with polar coordinates

Contour elements
- Undercut is not implemented

Contour processor:
- with coinciding solutions however, occasionally a selection dialogue appears
- occasionally an alternative solution (for starting point or end point) of a contour element cannot be selected. In this case a change to another contour element concerned (preceding, next element) is useful. Then the selection of the alternative solution can be carried out there.

Created code
- constant coordinate values are not generally omitted in the code (only with horizontal and/or vertical straight lines)
- selected multiple solutions cannot be retranslated from a Siemens code (the first solution is displayed)
- code with selected multiple solutions cannot be retranslated on an original control
- the text from the free input is deposited in an EMCO specific field but does not create an NC-code
- chamfers between straight line and circle and/or circle and circle create a different code

Error messages
- „Chamfer/radius too large"
  The value for the transition with chamfer or radius is too large.
  Remedy: select smaller transition value

EMCO

- „Inconsistent geometry values!"
  Due to the last input a contradiction for the contour element was noted.
  Examples:
  - Circle end points outside of circles
  - Intersection point at infinity
  - Tangents of points within a circle
  Remedy: correct the last input
- „Illegal function!"
  The last input or selection is not admissible at the moment.
  Examples:
  - Transitions to following element with radii with elements with tangential connection to the previous element
  - Cut-ins as transition (are generally not implemented)
  - Transitions at indefinite contour elments
  Remedy: select a valid transition to the following element
- „Make dialog selection first!"
  You want to make an input while waiting for the selection of a solution.
  Remedy: First make a selection, then further inputs can be carried out.

Help
- Help images are in a modal window, therefore the contour processor cannot be operated while a help image is displayed
- Not all help images of the original control are implemented

EMCO

D 106

E: Tool Correction / Tool Measuring

Tool Correction

Tool call

T..:   Tool number in magazine
D..:   Tool correction number

To every tool number T up to 9 correction numbers D can be assigned.

The control SINUMERIK 810D/840D describes the correction data D as edge.

A tool can have several correction numbers (e.g. a cut-in tool will be measured at the left and right corner).
Depending on usage in the program this tool is called e.g. with T1 D1 or T1 D2.

The command T..D.. activates the tool correction D and changes in the tool.
The data for the too correction (tool length, tool radius, ...) will be read from the tool data register.

Possible tool numbers:
T 1..32000, D 1..9

EMCO
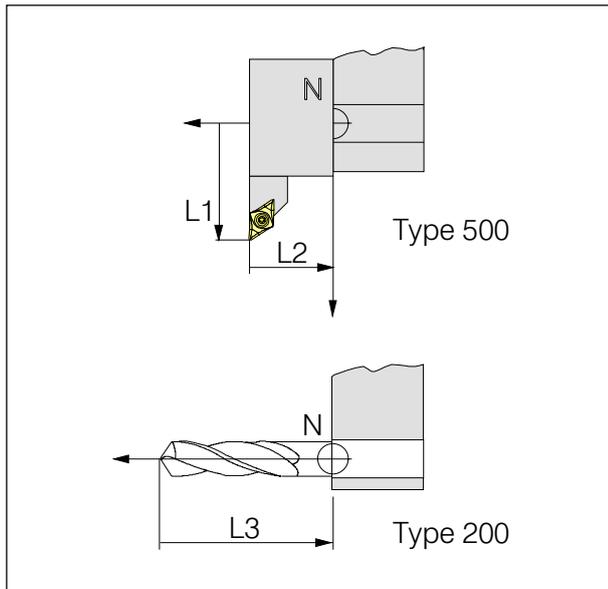
Direction of the length correction for the tool types

Tool length correction

The tool length correction L1 is effective vertical to the main plane (G17-G19).
Main application for turning: G18 - tool length correction L1 in X

The tool length correction shifts the tool zero from the tool mount reference point N to the tool tip.
By that all positions are in relation to the tool tip.

On most lathes the tool mount reference point N is on the face of the tool turret disk or the tool holder.
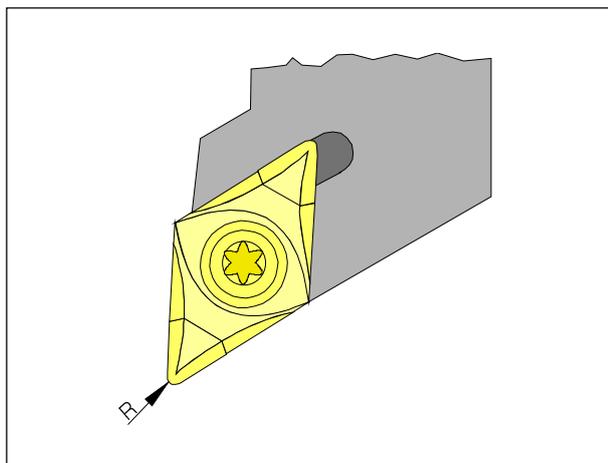
Cutter radius R

Cutter radius

Indicating a cutter radius is necessary only when a cutter radius compensation (G41, G42) is used for that tool.

EMCO
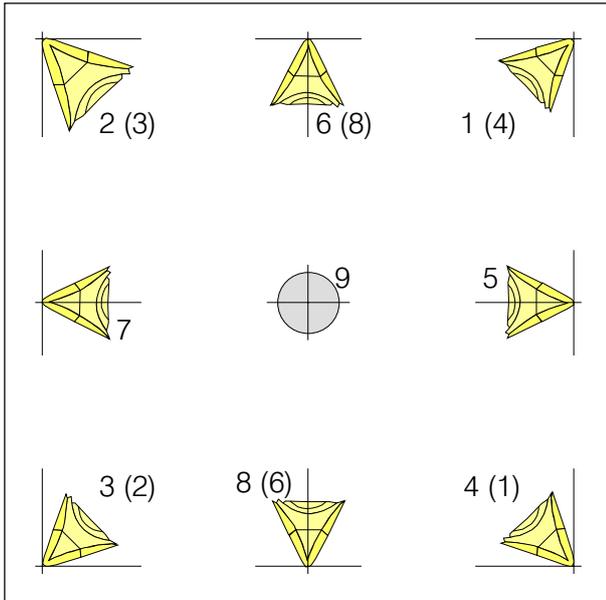
Cutter position of the tools

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.

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)

Tool types

Drilling tools

200	Twist drill
205	Solid drill
210	Boring bar
220	Centre drill
230	Countersink
231	Counterbore
240	Tap for regular threads
241	Tap for fine threads
242	Tap for Withworth threads
250	Reamer

Turning tools

500	Roughing tool
510	Finishing tool
520	Cut-in tool
530	Cut-off tool
540	Thread tool

EMCO

WinNC SINUMERIK 840D TURN (c) EMCO	_ □ X

Parameter	Channel 1	Jog	\PROG\MPF.DIR Part1.MPF	

Channel reset

Program aborted   ROV

T no. + ⇧F1

T no. - ⇧F2

Tool offsets   1

T-number	1	D-number	1	No. of c.edges	1
Tool type	510	Finishing tool			
C. edge pos.	3				

D no. + ⇧F3

D no. - ⇧F4

	Geometry	Wear	Base	
Tool length comp.				
Length 1 :				mm
Length 2 :				mm
:				
Radius compensation				
Radius :			mm	

Delete ⇧F5

go to ⇧F6

Technology
    Clear. angle :   Deg.
    DP25 res :

Overview ⇧F7

New ⇧F8

Tool offset F1	R parameter F2	Setting Data F3	Zero offset F4	User data F5	F6	F7	Determine compensa. F8 >

T number
With this number the tool will be called up (position number in tool turret)

D number
Number of the tool correction. A tool can have also several correction numbers (e.g. left and right corner of a cut-in tool).

No. of c. edges
Number of D numbers for the tool.

Tool type
This number determines the kind of tool.

Geometry
Dimensions of the tool

Wear
Deviation from the geometry value.

Base
Dimension of a tool holder, in which the tools will be clamped.

The sum of geometry, wear and base is the total effective tool correction.

emco

Softkeys:

T no +, T no -
Switches to the next higher or lower tool number.

D no +, D no -
Switches in the tool to the next higher or lower tool correction number.

Delete
Delete a tool from the list or delete a correction of the actual tool.
Press the softkey DELETE. The vertical softkey line shows the softkeys DELETE CUT. EDGE, DELETE TOOL and ABORT.

Delete tool
The actual tool and all its edges (corrections D) will be deleted.

Delete cutting edge
Always the cutting edge with the highes D number will be deleted.
The D numbers must be continuous without gap, e.g. a tool with four edges must have D1, D2, D3, D4 and only D4 can be deleted.
D1 can not be deleted, in this case the whole tool must be deleted (a tool must have at least one edge).

Abort
Exit without deleting.

Go to
Direct selection of tool.
Press the softkey GO TO. The vertical softkey list shows the selection softkeys and beside a input window.

Preselected tool
The number selected in a CNC program (while or after program run).

Active tool
The tool that is swivelled in in the tool turret.

Input field
Here you can enter the requested T and D number

and take over with ⏩ .

OK
Changes to the requested tool.

Abort
Exit without tool selection.

Overviews
Display of tool list.
Place the cursor on the requested tool and take over with softkey "OK".

New
Establish a new tool or a new corection (edge).

New tool edge
A correction data set will be added to an existing tool.
Enter the T number to which the new edge should be added (the actual tool is suggested) and the tool type of the new edge.

Confirm your input with ⏩ .

The softkey OK establishes the new edge, ABORT leaves without new edge.

New tool
A new tool will be added to the list.
Enter the T number and tool type of the new tool.

Confirm your input with ⏩ .

The softkey OK establishes the new tool, ABORT leaves without new tool.

EMCO

Tool Measuring

Scratch method

1 Clamp a face-machined workpiece with exact measured diameter.

2 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

3 Read and note the actual Z position.

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

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

6 Call tool data register
Operating Area Parameter - Tool Offset
Select the desired tool and the desired correction with the softkeys "T no.", "D no".

7 For drilling tools place the cursor on Geometry L3, for turning tools on Geometry L2.

8 Press softkey "Determine compensation".

9 In the field "Reference dimensions" set the axis on Z.

10 Enter the value from point 4 as "Reference value" in the field "Reference dimensions".

11 Take over the correction in Z with the softkeys "Include" and "OK".

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

13 For turning tools place the cursor on Geometry L1.

14 In the field "Reference dimensions" set the axis on X.

15 Enter the diameter of the workpiece as "Reference value" in the field "Reference dimensions".

16 Take over the correction in X with the softkeys "Calculation" and "OK".

17 Enter the remaining data (cutter radius, clearance angle, cutter position...).

18 Swivel in next tool, select T and D number and repeat from step 5, until all tools are measured.

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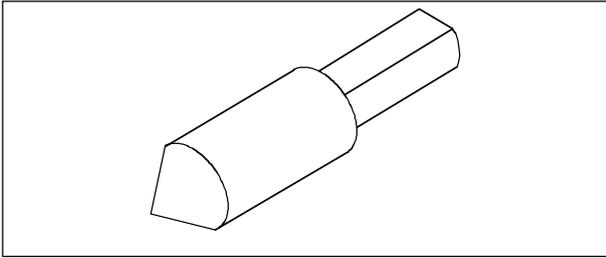
Reference tool Concept TURN 50/55
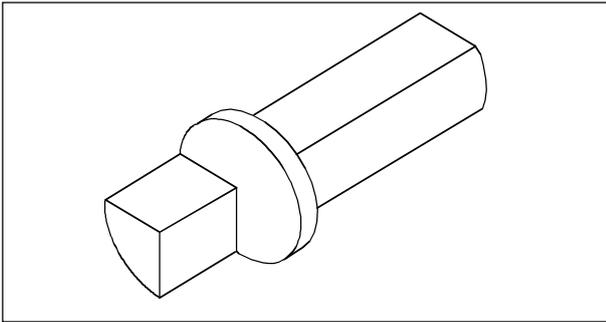
Reference tool Concept TURN 105/155
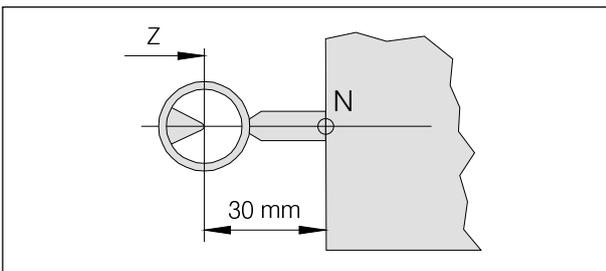
Reference tool measuring Concept Turn 50/55
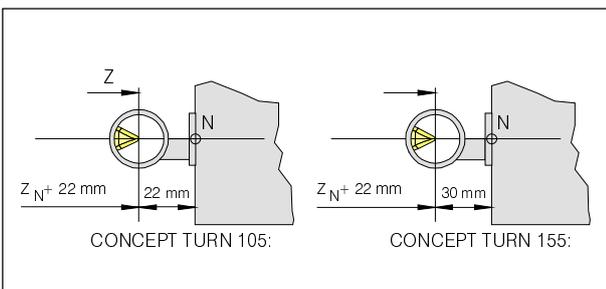
Reference tool measuring Concept Turn 105/155
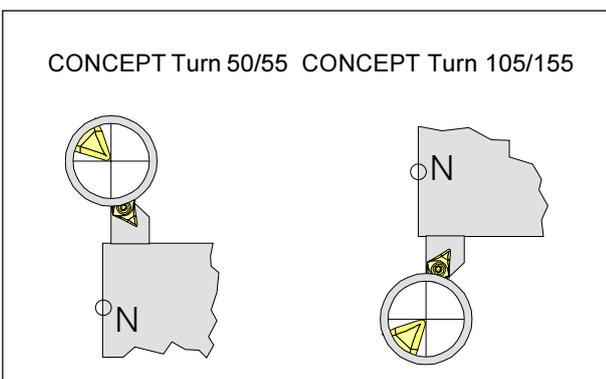
Traverse into the graticule with the tool

With optical presetting device

Principially same way as scratch method.
The optical way is more precise because touching will be avoided and the tool is displayed enlarged in the optics.

- 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.

- 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.

- In the menu "Parameter" - "Tool offset" - "Determine compensa" under reference value in X and Z put in the actual slide position
  (Z- value = Lengh of the reference tool)

- Swivel tool turret and traverse with the first tool to be measured into the reticule.
- Choose the T number and positioning the cursor on the axis.
  "Determine compensa", select axis and pull "Incluse".

- Clamp next tool etc.

emco

F: Program Run

Preconditions

Zero offsets G54-G57

The used zero offsets must be measured and entered.

Tools

The used tools must be measured and entered.
The tools must be at the corresponding positions (T) in the tool change system.

Reference point

The reference point must be approached in all axes.

Machine

The machine must be ready for operation.
The workpiece must be clamped safe.
Loose parts (clamping keys etc.) must be removed from the working area to avoid collisions.
The machine door must be closed for program run.

Alarms

No alarms must be active.

EMCO

Program Selection

Program overview

Operating Area Machine, Automatic mode.
Press the softkey PROGRAM OVERVIEW.

With the horizontal softkeys the programs of the specified type (workpieces, part programs, subprograms, standard cycles, user cycles, clipboard) can be displayed.

Program enable

To work off a program it must be enabled previous.
• Operating Area Machine, Automatic mode
• Press the softkey PROGRAM OVERVIEW
• Mark the desired program or workpiece with the keys .
• You can enable / disable the selected program / workpiece with the softkey ALTER ENABLE.
• Enable is displayed with an (X) in the list:
  (X) enabled
  ( ) disabled
• When a program is part of a workpiece (directory), the program and the workpiece must be enabled.

Select program for working off

Select program
• Operating Area Machine, Automatic mode
• Press the softkey PROGRAM OVERVIEW

• Mark the program with the keys .
• Press the softkey PROGRAM SELECTION.
• The program name is displayed right in the headline.

Select workpiece
• In the workpiece overview select a workpiece with the keys .
• Press the softkey WORKPIECE SELECTION.
• If there exists a part program with the same name in this directory, it will be selected for machining automatically (e.g. with selection of the workpiece PART1.WPD the part program PART1.MPF will be selected automatically).
• The program name and the workpiece information is displayed right in the headline.
• If there exists a initialisation file with the same name in this directory, it will be executed immediately with selection of the part program (e.g. PART1.INI).

EMCO

Program Start, Program Stop

Select a program for machining.

Change into Operating Area Machine, Automatic mode.

Press the key [⟨|⟩] for program start.

Stop program with [⃝], continue with [⟨|⟩].

Abort program with [⫽].

Messages while program run

3 Stop: Emergency stop active
The EMERGENCY OFF button was pressed.

4 Stop: Alarm active with stop
An alarm stopped the program.

5 Stop: M0/M1 active
Programmed stop of program run.

Continue with [⟨|⟩].

6 Stop: Block ended in SBL mode
A block was finished in single block mode.

Continue with [⟨|⟩].

7 Stop: NC Stop active

The program was stopped with the key [⃝].

Continue with [⟨|⟩].

8 Wait: Read-in enable missing
Read-in enable is a signal from the machine to the control.
The actual block has not been worked off yet (e.g. tool change, dividing device, bar loader, etc.).
The next program block will be worked off only after the previous was finished.

9 Wait: Feedrate enable missing
Feed enable is a signal from the machine to the control.
The actual block has not been worked off yet (e.g. spindle did not yet reach the programmed speed, etc.).
The next program block will be worked off only after the previous was finished.

10 Wait: Dwell time active
Program run was stopped for the programmed dwell time.

17 Wait: Feedrate override to 0%
The feed override switch is on 0% position.

18 Stop: NC block incorrect
Programming error

21 Wait: Block search active
While block search all blocks before the search target will be simulated internally first and at the search target machining will start.

EMCO

Program Control

Press the softkey PROGRAM CONTROL.

Select the desired function with the cursor keys $\boxed{\text{\$ } \downarrow}$

and $\boxed{\text{! } \uparrow}$.

Activate / disactivate the function with the key $\boxed{\text{> } \circlearrowleft}$.

SKIP Skip block
When this function is active, all blocks that are signed with a slash before the block number (/N...) will not be executed while program run.

DRY Dry run feedrate
For test run without workpiece (no machining).
All blocks with a programmed feedrate (G1, G2, G3, G33, ...) will traverse with the predefined dry run feed instead of the programmed feedrate.
The spindle does not run.

ROV Rapid traverse override

SBL1 Single block with stop after machine function blocks
Program run will be stopped after every movement.

Continue with $\boxed{\lozenge}$.

SBL2 Single block with stop after every block
Program run will be stopped after every block, also when no movement is programmed in the block (calculating block).

M01 Programmed stop
With M01 in the program the program normally will not stop at this command.
When this function is active, the program stops at M01.

Continue $\boxed{\lozenge}$.

DRF DRF selection
Additional incremental zero offset with the electronic handwheel.

PRT Program test
Program test without axis movement.

EMCO

Block Search

Block search allows to run a program forward until a required block and then start machining.
Two types of block search are available.

1. With calculation at the contour
While block search the same calculations are preceeded as with a normal program run (the program is simulated internally).

At the block start of the search block that machine status will be established, that would be active also with normal program run. Afterwards the search block will be worked off like a normal program run block.

2. With calculation at the block end point
While block search the same calculations are preceeded as with a normal program run (the program is simulated internally).

At the block end of the search block that machine status will be established, that would be active also with normal program run. The block end of the search block will be approached direct, the search block itself will not be worked off.

Sequence:

• Operating Area Machine, AUTO mode is selected.
• The program for block search is selected.
• The control is in RESET status.

• Press the softkey BLOCK SEARCH.
• Place the cursor on the search block.
• Block search will be started with the softkey CALCULATE CONTOUR or CALCULATE BLK ENDPT.
• The control calculates all blocks until search target but executes no movements.

• ▨ aborts block search.

• ◁▷ starts the program run. The screen shows a

security query. Confirm it with ◁▷.

• The position for the search target will be approached with a compensation movement and from that time on the program runs automatically.

EMCO

F 6

G: Flexible NC- Programming

Variable and arithmetic parameters

Variables can be used instead of fixed values to increase the flexibility of a program. You can respond to signals such as measured values or, by storing setpoints in the variables, you can use the same program for different geometries.
A skilled programmer can use variable calculation and programm jumps to create a highly flexible program archive which will considerably reduce the programming work required.

Variable types

- User defined variables
- Arithmetic parameters
- System variables

Variable types

INT
Integers with leading sign
Value range: $\pm(2^{31} - 1)$

REAL
Fractions with decimal point
Value range: $\pm(10^{-300} ... 10^{+300})$

BOOL
Boolean values: TRUE (1) and FALSE (0)
Value range: 1, 0

CHAR
1 ASCII character spezified by the code
Value range: 0 ... 255

STRING
Character string, number of character in [...],
maximum 200 Characters
Value range: Folge von Werten mit 0 ... 255

AXIS
Axis names (Axis adresses) only
Value range: All axis identifiers and spindles of a channel.

FRAME
Geometrical parameters for translation, rotation, scalling, mirroring.

System variable

Variables provided by the control wich can be processed in the program. system variables provide access to zero offsets, tool offset, actual values, measured values on the axes, control states, etc.
System variablees return values of the defined type. Some of the system variables cannot be assigned values.
The name of a system variable always identified by the "$" character followed by the specific names.

Overview of the system variable types

1st letter	Meaning
$M	Machine data
$S	Setting data
$T	Tool management data
$P	Programmed values
$A	Current values
$V	Service data
2nd letter	Meaning
N	NCK- global
C	Channel- specific
A	Axis- specific

Example:

$AA_IM ..... Current axis specific value in the machine coordinate system.

EMCO

Variable definition

User defined variables

In addition to the predefined variables, the programmer can also define his own variables and assign values to them.
Local variables are only valid in the ptogram in which they are defined.
Global variables apply in all programs.

Variable name

A variable name consists of up to 32 characters. The first two characters must be a letter or an underscore. The "$" character cannot be used for user defined variables, as it is reserved for system variables.

Format:

DEF INT name
or DEF INT name=Value

DEF REAL name
or DEF REAL name1, name2=3, name4
or DEF REAL name [array index1, array index2]

DEF BOOL name

DEF CHAR name
or DEF CHAR name [array index]=("A","B",...)

DEF STRING [string length] name

DEF AXIS name
or DEF AXIS name [array index]

DEF FRAME name

If a value is not assigned to a variable when it is defined, the system initializes it with zero.
Variables must be defined at the beginning of the program before use.
The definition must be made in a seperate block.
Only one variable type can be defined per block.

Example

Variable type INT	
DEF INT NUMBER	A variable type integer is created with the name NUMBER. The system initializes the variable with zero.
DEF INT NUMBER=7	A variable type integer is created with the name NUMBER. The system initializes the variable with the value 7.

Variable type REAL	
DEF REAL DEPTH	A variable type Real is created with the name DEPTH. The system initializes the variable with zero.
DEF REAL DEPTH=6,25	A variable type Real is created with the name DEPTH. The initial value is 6,25.

Variable type BOOL	
DEF BOOL IF_TOOMUCH	A variable of type Bool is created with the name IF_TOOMUCH. The system initializes the variable with zero (FALSE)
DEF BOOL IF_TOOMUCH=1 DRF BOOL IF_TOOMUCH=TRUE DEF BOOL IF_TOOMUCH=FALSE	A variable of type Bool is created with the name IF_TOOMUCH.

Variable type CHAR	
DEF CHAR GUSTAV_1=65	You can assign a code for the ASCII character to the variable of type Char or assign the ASCII character directly (65 is the code for the letter A)
DEF CHAR GUSTAV_1="A"	

Variable type STRING	
DEF STRING [6]SAMPLE_1="START"	Variables of type String can store a string of characters. The minimum number of characters is enclosed in square brackets after the variable type.

Variable type AXIS	
DEF AXIS AXISNAME=(X1)	The variables of type Axis have the name Axisname and contain the axis identifier of a channel, here X1

A varible of type AXIS stores names and spindle identifiers of a channel.
Axis names with extended addresses must be enclosed in parentheses.

EMCO

Array definition

Programming

DEF CHAR NAME[n,m]
DEF INT NAME[n,m]
DEF REAL NAME[n,m]
DEF AXIS NAME[n,m]
DEF FRAME NAME[n,m]
DEF STRING[string length] NAME[m]
DEF BOOL [n,m]

INT NAME[n,m]   Variable type (CHAR, INT,
                    REAL, AXIS, FRAME, BOOL)

DEF STRING[string length] NAME[m]
                    The data type STRING can only
                    be defined with one dimensio-
                    nal arrays

NAME   Variable name

Arrays with a maximum of 2 dimensions can be
defined.
Arrays with STRING variables may only be one
dimensional. The string lenght is spezified after the
data type String.
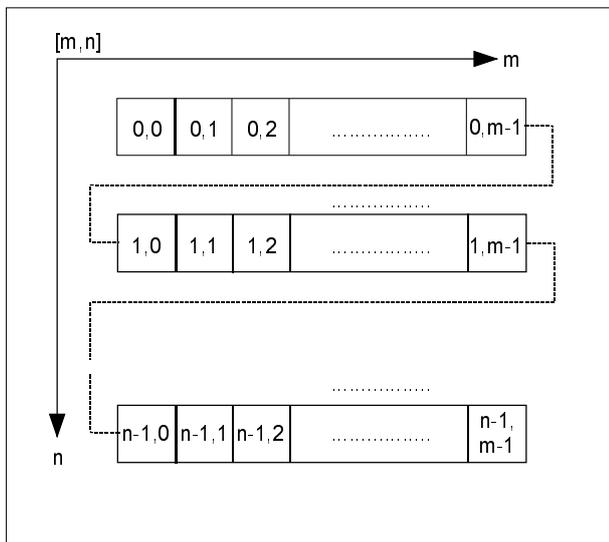
Array index

The elements of an array can be accessed via the
array index.
The array elements can either be read or assigned
values using this array index.
The first array elements begins with the index [0,0].
With an array size of [3,4], for example, the maximum
array index is [2,3].

In the marginal example, the initialization values
match the index of the array element in order to
illustrate the order of the individual array elements.

Initialization of arrays

Initialization values can be assigned to arrays
elements during program execution or when arrays
are defined.
The rigtht hand array index is incremented first on
two dimensional arrays.

EMCO

Initialization of value lists, SET

Options during array definition

DEF Typ VARIABLE=SET(Value)
DEF Typ ARRAY[n,m]=SET(Value,Value,...)
oder
DEF Typ VARIABLE=Value
DEF Typ ARRAY[n,m]=(Value,Value,...)

- The number of array elements assigned corresponds to the number of initialization values programmed.
- Array elements without values are automatically assigned the value "0".
- There may be no gabs in the value list for variables of the AXIS type.
- If more values are programmed than remaining array elements exist, the system trigger an alarm.

Options during program execution

ARRAY[n,m]=SET(Value, Value,...)
ARRAY[n,m]=SET(Expression, Expression,...)

- Field elements are initialized as described above for array definition.
- Expressions may also be used here as initialization values.
- Initialization starts at the programmed array indices. Values can also be assigned selectiely to subarrays.

Example

Assignment of expressions
DEF INT ARRAY[5,5]
ARRAY[0,0]=SET(1,2,3,4,5)
ARRAY[2,3]=SET(Variable,4*5.6)

The axis index is not processed for axis variables.

Example

Initialization on one line
$MA_AX_VELO_LIMIT[1,AX1]=SET(1.1,2.2,3.3)

Corresponds to:

$MA_AX_VELO_LIMIT[1,AX1]=1.1
$MA_AX_VELO_LIMIT[2,AX1]=2.2
$MA_AX_VELO_LIMIT[3,AX1]=3.3

Initialization with identical values, REP

Options during array definition

DEF Typ ARRAY[n,m]=REP(Value)

All array elements are assigned the same value (constant).

> Variables of type FRAME cannot be initialized.

Example

DEF REAL ARRAY5[10,3]=REP(9.9)

Options during program execution

ARRAY[n,m]=REP(value)
ARRAY[n,m]=REP(expression)

- Expressions may also be used here as initialization values.
- All array elements are initialized with the same value.
- Initilization starts at the programmed array indices. Values can also be assigned selectively to subarrays.

> Variables of the FRAMe type are permitted and can be initialized very simple using this method.

Example

Initialization of all elements with one value.

DEF FRAME FRM[10]
FRM[5]=REP(CTRANS(X,5))

EMCO

Example

Initialization of complete variable arrays.
The drawing shows the current assignment.

N10 DEF REAL ARRAY1 [10, 3] = SET(0, 0, 0, 10, 11, 12, 20, 20, 20, 30, 30, 30, 40, 40, 40,)
N20 ARRAY1 [0,0] = REP (100)
N30 ARRAY1 [5,0] = REP (-100
N40 ARRAY1 [0,0] = SET (0, 1, 2, -10, -11, -12, -20, -20, -20, -30, , , , -40, -40, -50, -60, -70)
N50 ARRAY1 [8,1] 0 SET (8.1, 8.2, 9.0, 9.1, 9.2)

[1,2]	N10: Initialization with defination			N20/N30: Initialization with identical value			N40/N50: Initialization with different values		
	0	1	2	0	1	2	0	1	2
0	0	0	0	100	100	100	0	1	2
1	10	11	12	100	100	100	-10	-11	-12
2	20	20	20	100	100	100	-20	-20	-20
3	30	30	30	100	100	100	-30	0	0
4	40	40	40	100	100	100	0	-40	-40
5	0	0	0	-100	-100	-100	-50	-60	-70
6	0	0	0	-100	-100	-100	-100	-100	-100
7	0	0	0	-100	-100	-100	-100	-100	-100
8	0	0	0	-100	-100	-100	-100	8.1	8.2
9	0	0	0	-100	-100	-100	9.0	9.1	9.2
	The array elements [5,0] to [9,2] have been initialized with the default value (0.0).						The array elements [3,1] to [4,0] have been initialized with the default value (0.0). The array elements [6,0] to [(,0] have not been changed		

2

1

EMCO

Indirect programming

Indirect programming enables programs to be used universally. Tha extended address (index) is substituted by a variable of suitable type.

All addresses can be configured, except for::
- N- block number
- G- G command
- L- subprogram

Indirect programming is not possible for any settable addresses. (X[1] is not permitted instead of X1).

Example

S1=300   Direct programming

DEF INT SPINU=1
S[SPINU]=300

                    Indirect programming:
                    Speed 300rpm for the spindle whose number is stored in the variable SPINU.

Assignments

Values of matching types can be assigned to variables/arithmetic parameters in the program.
The assignment is always made in a seperate block.
Up to two assignments are possible per block.
Assignments to axis addresses always require a seperate block to variable assignments.

Example

R1=10.518 R2=4 Vari1=45
X=47.11 Y=R2

R1=R3 VARI1=R4   Assignment of
                        numeric value

R4=-R5 R7=-VARI8   Assignment of a variable of matching type
                        Assignment of opposite leading sign (only allowed with types INT/REAL).

Assignment to string variables

A distinction is made between upper an lower case characters within a CHAR or STRING.

Example

MSG("Finishin contour")
Displays the text "Finishing contour" on the screen.

EMCO

Arithmetic operations/functions

Arithmetic functions are used predominantly for R parameters and variables of the type REAL. The types INT and CHAR are also permitted

Standard mathematical notation are used in the arithmetic operations. Priorities for execution are indicated by parentheses. Angles are specified for trigonometry functions and their inverse functions (right angle = 90°).

COMMAND	MEANING
+, -, *, /	Arithmetic functions
SIN()	Sine
COS()	Cosine
TAN()	Tangent
ASIN()	Arcsine
ACOS()	Arccosine
ATAN2()	Arctangent2
SQRT()	Square root
SQR()	2nd potency
ABS()	Absolute number
TRUNC()	Truncate to integer
ROUND()	Rounding
POT()	2nd power (square)
LN()	Natural logarithm
EXP()	Exponential- Function

Arithmetic function, ATAN2(,)

The function calculates the angle of the resulting vector from two vectors at right angles to each other. The result is in one of four quadrants (–180° < 0 < +180°). The angular reference is always based on the 2nd value in the positive direction.

Example

R1=R1+1   new R1 = old R1 +1
R1=R2+R3 R4=R5-R6 R7=R8*R9
R10=R11/R12 R13=SIN(25.3)
R14=R1*R2+R3   Multiplication and division have priority over addition and subtraction
R14=(R1+R2)*R3   Parentheses are calculated first
R15=SQRT(POT(R1)+POT(R2)) Inner parentheses are solved first.
                         R15 = Square root of $(R1^2 + R2^2)$.
RESFRAME= FRAME1:FRAME2
FRAME3=CTRANS(...):CROT(...)   The chain operator combines frames in a resulting frame or assigns values to the frame components.

Comparison and logic operations

Comparison operators

The comparison operators can be used for variables of type CHAR, INT, REAL and BOOL.
The code value is compared with the CHAR type.
The following are possible with types STRING, AXIS and FRAME: == und <>.
The result of a comparison operation is always type BOOL.
Comparison operations can be used, for example, to formulate a jump condition.

==	Equal to
<>	Not equal to
>	Greater than
<	Less than
>=	Greater than or equal to
<=	Less than or equal to
<<	Chaining of strings

Example

IF R10>=100 GOTOF DEST
oder
R11=R10>=100
IF R11 GOTOF DEST
The result of the comparison R10>=100 is first buffered in R11.

Logic operators

Logic operators are used to logically combine truth values. AND, OR, NOT and XOR can generally only be used on variables of type BOOL, however, they can also be used on the data types CHAR, INT and REAL by means of iplicit type conversion.
Spaces must be inserted between Boolean operands and operatoers.
In logic (Boolean) operations the following applies to the data types BOOL, CHAR, INT and REAL:
0 is equivalent to FALSE
not equal to 0 is equivalent to TRUE

AND	AND
OR	OR
NOT	NOT
XOR	Exklusiv OR

Parentheses can be used in arithmetic expressions to define the order of execution for all operators and thus to override the normal priority rules.

IF (R10<50) AND ($AA_IM[X]>=17.5)GOTOF DEST
IF NOT R10 GOTOB START

Bit operators

Bit for bit logic operations can also be performed on variables of the type CHAR and INT. Type conversion takes place automatically.

B_AND	Bit AND
B_OR	Bit OR
B_NOT	Bit NOT
B_XOR	Bit exclusive OR

The operator B_NOT refers only to an operand; this follows the operator.

Example

IF $MC_RESET_MODE_MASK B_AND 'B10000' GOTOF ACT_PLANE

EMCO

Priority of operators

Each operator is assigned a priority. When an expression is evaluated, the operators with the highest priority are always applied first. Where operators have the same priority, the evaluation is from left to right.

Paratheses can be used in arithmetic expressions to define the order of execution for all operators and thus to override the normal priority rules.

Priority of operators (highest to lowest)

1. NOT, B_NOT	Negation, bit negation
2. *, /, DIV, MOD	Multiplication, division
3. +, −	Addition, subtraction
4. B_AND	Bit AND
5. B_XOR	Bit exclusive OR
6. B_OR	Bit OR
7. AND	AND
8. XOR	exclusive OR
9. OR	OR
10. <<	Chaining of strings, result type STRING
11. ==, <>, >, <, >=, <=	Comparision operators

The chain operator „:" for frames may not appear with other operators in an expression. A priority level is this not required for this operator.

Type conversion

The constant numeric value, variable or expression assigned to a variable must be compatible with the type of this variable. If this is this case, the type is automatically converted when the value is assigned.

Possible type conversion

from \ to	REAL	INT	BOOL	CHAR	STRING	AXIS	FRAME
REAL	yes	yes*	yes[1]	yes*	-	-	-
INT	yes	yes	yes[1]	yes[2]	-	-	-
BOOL	yes	yes	yes	yes	yes	-	-
CHAR	yes	yes	yes[1]	yes	yes	-	-
STRING	-	-	yes[4]	yes[3]	yes	-	-
AXIS	-	-	-	-	-	yes	-
FRAME	-	-	-	-	-	-	yes

*   On type conversion from REAL to INT, a fraction >=0.5 is rounded up, otherwise this fraction is roundet down (same effect as ROUND function)

1) Values <> 0 are TRUE, Values == 0 are FALSE
2) If the value is in the permitted value range
3) If onlx 1 character
4) String runs 0 = >FALSE, otherwise TRUE

If a value is greater than the target range on conversion, an error message is generated.

If mixed types occur in an expression, a type conversion is performed automatically.

EMCO

Lenght of strings, STRLEN

This functionality allows the lenght of a string to be specified.

Syntax:

INT_ERG = STRLEN (STRING)	Result type: INT

Semantics:

A number of characters is returned that - counting from the beginning of the string- are not 0 characters.

Example:

This function can be used to determine the end of the string, for example, in connection with the single character access described below:

IF(STRLEN(BAUSTEIN_NAME)>10)GOTOF FEHLER

CASE statement

Format:

CASE (expression) OF constant1 GOTOF LABEL1 DEFAULT GOTOF LABELn
CASE (expression) OF constant1 GOTOB LABEL1 DEFAULT GOTOB LABELn

CASE   Vocabulary word for jump instruction
GOTOF   Jump instruction with jump destination forwards
GOTOB   Jump instruction with jump destination backwards
LABEL   Destination (label within the program)
LABEL: The name of the jump destination is followed by a colon
Expression arithmetic expression
Constant   Constant of type INT
DEFAULT   Program path if none of the previously named constants applies

The CASE statement enables various branches to
be executed according to a value of type INT.

The program jumps to the point specified by the jump
destination, depending on the value of the constant
evaluated in the CASE statement.

In cases where the constant matches none of the
predefined values, the DEFAULT instruction can
be used to determine the jump destination.
If the DEFAULT instruction is not programmed,
the jump destination is the block following the
CASE statement.

CASE(expression) OF 1 GOTOF LABEL1 2 GOTOF LABEL2 ... DEFAULT
GOTOF LABELn
„1" and „2" are possible constants.
If the value of the expression = 1 (INT-constant), jump to block with LABEL1
If the value of the expression = 2 (INT-constant), jump to block with LABEL2
…
otherwise jump to the block with LABELn

Example

DEF INT VAR1 VAR2 VAR3
CASE(VAR1+VAR2-VAR3) OF 7 GOTOF LABEL1 9 GOTOF LABEL2 DEFAULT
GOTOF LABEL3
LABEL1: G0 X1 Y1
LABEL2: G0 X2 Y2
LABEL3: G0 X3 Y3

EMCO

Check structures

IF-ELSE-ENDIF .... Selection between 2 alter-
natives
LOOP-ENDLOOP .... Endless loop
FOR-ENDFOR .... Count loop
WHILE-ENDWHILE ..... Loop with condition at
beginning of loop
REPEAT-UNTIL .... Loop with condition at end
of loop

The control processes the NC blocks as standard in the programmed sequence.
In addition to the program branches described in this Section , these commands can be used to define additional alternatives and program loops.
These commands enable the user to produce wellstructured and easily lrgible programs.

IF-ELSE-ENDIF
An IF-ELSE-Endif- block is used to select one of two alternatives:
IF (expression)
N50...
N60...
ELSE
N120...
ENDIF

If the value of the expression is TRUE, i.e. the condition is fulfilled, then the next program block is executed. If the condition is not fulfilled, then the ELSE program branch is executed. THe ELSE branch can be omitted.

Endless- Program loop, LOOP

Endless loops are used in endless programs. At the end of the loop, there is always a branch bach to the beginning.

LOOP
N50...
N60...
ENDLOOP

Count loop, FOR
The FOR loop is used if it is necessary to repeat an operation by a fixed number of runs. The variable must be of the INT type.

FOR Variable = start value TO endvalue
N50...
N60...
ENDFOR

EMCO

Program loop with condition at beginning of loop, WHILE

The WHILE program loop is executed for as long as the condition is fulfilled.

WHILE expression
N50...
N60...
ENDWHILE

Program loop with condition at the end of loop, REPEAT

The REPEAT loop is executed once and repeated continuously until the condition is fulfilled.

REPEAT
N50...
N60...
UNTIL(expression)
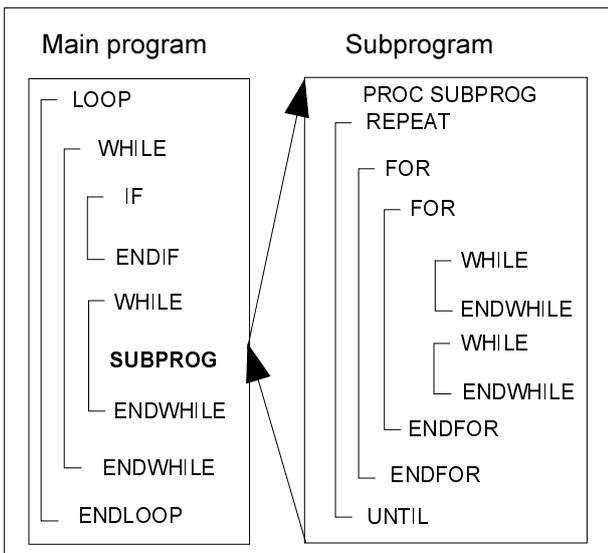
Nesting depth

Check structures apply locally within programs. A nesting depth of up to 8 check structures can be set up on each subprogram level

```
┌─ Main program ──────┐   ┌─ Subprogram ────────┐
│                     │   │                     │
│ ┌─ LOOP            ╱│───│→  PROC SUBPROG       │
│ │                 ╱ │   │   ┌─ REPEAT          │
│ │ ┌─ WHILE       ╱  │   │   │                  │
│ │ │ ┌─ IF       ╱   │   │   │ ┌─ FOR           │
│ │ │ │          ╱    │   │   │ │ ┌─ FOR         │
│ │ │ └─ ENDIF  ╱     │   │   │ │ │              │
│ │ │          ╱      │   │   │ │ │ ┌─ WHILE     │
│ │ ┌─ WHILE ╱        │   │   │ │ │ └─ ENDWHILE  │
│ │ │       ╱         │   │   │ │ │ ┌─ WHILE     │
│ │ │ SUBPROG ◄──────╲│   │   │ │ │ └─ ENDWHILE  │
│ │ └─ ENDWHILE      ╲│   │   │ │ └─ ENDFOR      │
│ │                   │   │   │ │                │
│ └─ ENDWHILE         │   │   │ └─ ENDFOR        │
│                     │   │   │                  │
│ └─ ENDLOOP          │   │   └─ UNTIL           │
│                     │   │                     │
└─────────────────────┘   └─────────────────────┘
```

Runtime response

In interpreter mode (active as standard), it is possible to shorten program processing times more effectively by using program branches than can be opatined with check structures.
There is no difference between program branches and check structures in precompiled cycles.

EMCO

Supplementary conditions

Blocks with check structures elements cannot be suppresed. Labels may not be used in blocks of this type.

Check structures are processed interpretively.
When a loop end is detected, a search is made for the loop beginning, allowing for the check structures found in the process.

For this reason, the block structures of a program is not checked completely in interpreter mode.

If is not generally advisable to use a mixture of check structures and program branches.

A check can be made to ensure that check structures are nested correctly when cycles are preprocessed.

Check structures may only be inserted in the statement section of a program. Definitions in the program header may not be executed conditionally or repeatedly.

It is not permissible to superimpose macros on vocabulary words for check structures or on branch destinations. No such check is made when the macro is defined.

Example
(Endless program)

```
%_N_LOOP_MPF
LOOP
   IF NOT $P_SEARCH              ;no block search
      G01 G90 X0 Z10 F1000
      WHILE $AA_IM[X] <= 100
         G1 G91 X10 F500          ;Drilling pattern
         Z–5 F100
         Z5
         ENDWHILE
         Z10
   ELSE                          ; Block search
MSG(„No drilling during block search")
   ENDIF
   $A_OUT[1] = 1                 ;next drilling plate
   G4 F2
ENDLOOP
M30
```

Example (Production of a fixed quantity of parts)
```
%_N_WKPCCOUNT_MPF
DEF INT WKPCCOUNT
FOR WKPCCOUNT =0 TO 100
G01 …
ENDFOR
M30
```

EMCO

Suppress current block display, DISPLOF, DISPLON

Format

PROC ... DISPLOF

With DISPLOF the current block display for a subprogram is suppressed. DISPLOF is placed at the end of the PROC statement.
Instead of the current block, the call of the cycle or the subprogram is displayed.

By default the block display is activated. Deactivation of block display with DISPLOF applie until the return from the subprogram or end of program. If further subprograms are called from the subprogram with the DISPLOF attribute, the current block display is suppressed in these as well. If a subprogram with suppressed block display is interrupted by an asynchronous subprogram, the blocks of the current subprogram are displayed.

**Single set suppression
SBLOF, SBLON**

Format

PROC ... SBLOF
PROC ... SBLON

SBLOF .... Single set suppression OF
SBLON .... Single set suppression ON

Single set suppression program specific
With SBLOF qualified programs are worked out as a complete block.

Example:
PROC Example SBLOF
G1 X10
RET

Single set suppression at the program
SBLOF can stand alone in the block. From this block single block is switched off until the next SBLON or until the end of the active subroutine level.

EMCO
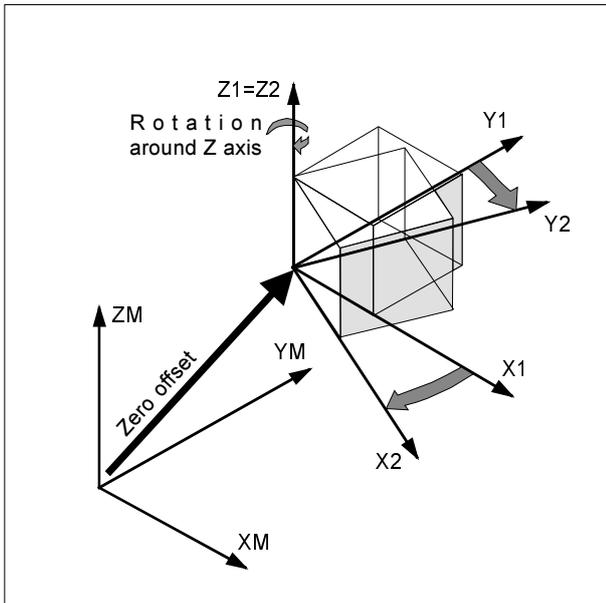
Frames

Frame is the conventional term for a geometrical expression that describes an arithmetic rule, such as translation or rotation.

Frames are used to describe the position of a destination coordinat system by specifying coordinates or angels starting from the current workpiece coordinate system.

Possible frames:

*   Basis frames (basis offset)
    Describe the transformation of coordinates from the basis system of coordinates (BKS) into the zero point system (BOS) and have the same effect as adjustable frames.
*   adjustable frames (G54...G599)
    Adjustable frames are zero point offsets to be called up from any NC program with the commands G54 to G599. The offset values are preset by the operator and stored in the zero point memory of the control.
*   programmable frames.
    Programmable frames (TRANS, ROT,..) are valid in the actual NC-program and refer to the adjustable frames. They are used to determine the workpiece coordinate system.
*   additive frames
    The actually adjusted workpiece zero point or the one programmed last via frames serves as reference.

Frame variable/ frame relationship

Using the frames described above, arithmetic rules are specified to describe how the positions of the coordinate systems are releated
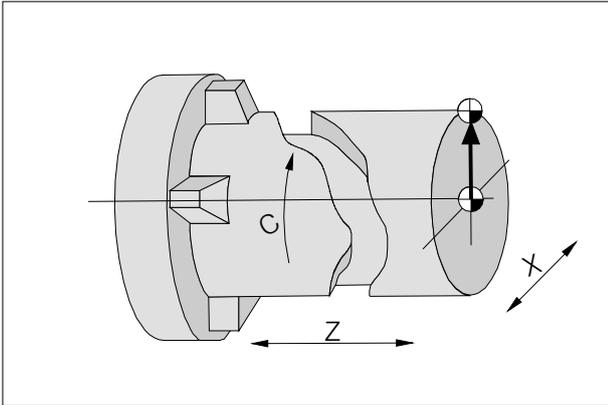
Frame variable:
$P_PFRAME means: current programmable frame.

Predefined frame variables

$P_IFRAME
Current settable frame variable that sets up the reference between the machine and workpiece coordinate systems.

$P_IFRAME contains the translation or rotation, etc., defined by G54.
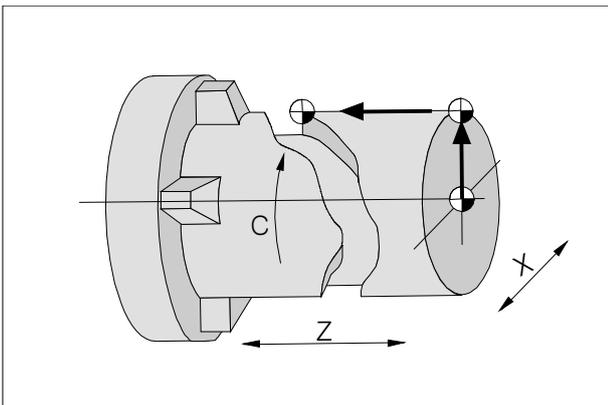
$P_BFRAME
Actual basis frame variable which establishes the reference between the basis system of coordinates and the basis zero point system which can be defined by the operator.

$P_PFRAME
Actual programmable frame variable which establishes the reference between workpiece zero point system and the workpiece coordinate system.

$P_PFRAME contains the resulting frame, which results from programming TRANS/ATRANS, ROT/AROT, SCALE/ASCALE, MIRROR/AMIRROR to the programmable frame.
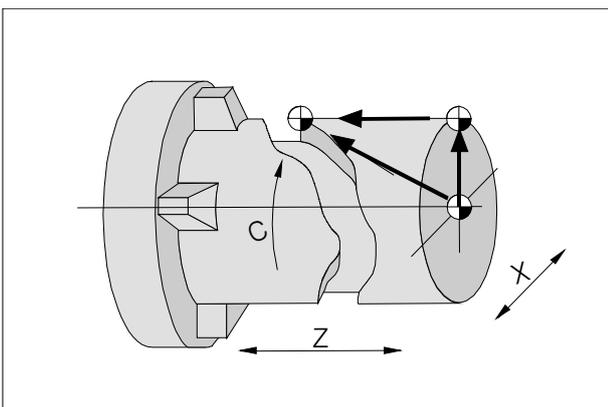
$P_ACTFRAME
Actual, resulting total frame consisting of the actual basis frame variable $P_BFRAME, the actual adjustable frame variable $P_IFRAME and the actual programmable frame variable $P_PFRAME.

$P_ACTFRAME describes the actually valid workpiece zero point.

Predefined adjustable frames $P_UBFR[n]

Writing on the predefined frame variable $P_UBFR[n] does not activate the basis frame at the same time but the activation is carried out only with the first execution of a G500, G54,... G599 instruction.

Predefined adjustable frames $P_UIFR[n]

The adjustable zero point offsets G54 until G599 can be read or written from the workpiece program by means of the predefined frame variable $P_UIFR[n].

5 adjustable frames ($P_UIFR[0] until $P_UIFR[4]) and/or 5 synonymous G commands - G500 (switch-off) and G54 until G57 - are preset as a standard .

$P_UIFR[0] corresponds G500
$P_UIFR[1] corresponds G54
$P_UIFR[2] corresponds G55
$P_UIFR[3] corresponds G56
$P_UIFR[4] corresponds G57

A total of 100 systems of coordinates can be created by predefined adjustable frames, which can be programmed to overlap the program e. g. as zero point for various devices.

EMCO

Axis function AXNAME, ISAXIS, AX

AXNAME("TRANSVERSE AXIS")
AX[AXNAME("STRING")]
SPI(spindle number)
ISAXIS(geometry axis number)

AXNAME ... Converts an input string to an axis identifier.
AX .... Variable axis identifier.
ISAXIS .... Checks whether the specified geometry axis exists

AXNAME
Is used, for example, to create generally applicable cycles when the name of the axes are not known.

ISAXIS
Is used in universal cycles in order to ensure that a specific geometry axis exists and thus that any following $P_AXNX- call is not aborted with an error message.

EMCO

DIAMON, DIAMOF

Format
DIAMON
DIAMOF

DIAMON   Diameter as dimension
DIAMOF   Radius as dimension (initial setting)

At DIAMON/DIAMOF you can choose between diameter as dimension or radius as dimension.

After activating DIAMON, diameter dimensions are defined for the specified transverse axis.
Diameter values apply to the following data:

- Actual value display of transverse axis in the workpiece coordinate system
- Programming:
  Final positions, independent of G90/G91 interpolation parameters with G2/G3, if they are programmed absolutely with AC.
- Read actual values in the workpiece coordinate system with MEAS, MEAW, $P_E[X], $AA_IW[X]

By programming DIAMOF you can switch at any time to radius as dimension.

EMCO

H: Alarms and Messages

Machine Alarms 6000 - 7999

These alarms will be triggered by the machines. There are different alarms for the different machines.

The alarms 6000 - 6999 normally must be confirmed with RESET. The alarms 7000 - 7999 are messages which normally will disappear when the releasing situation is finished.

PC MILL 50 / 55 / 100 / 105 / 125 / 155
Concept MILL 55 / 105 / 155

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

6001: PLC-CYCLE TIME EXCEEDING
Contact EMCO Service.

6002: PLC - NO PROGRAM CHARGED
Contact EMCO Service.

6003: PLC - NO DATA UNIT
Contact EMCO Service.

6004: PLC - RAM MEMORY FAILURE
Contact EMCO Service.

6005: OVERHEAT BRAKEMODUL
Main drive was braked too often, large changes of speed within a short time. E4.2 active

6006: OVERLOAD BRAKE RESISTOR
see 6005

6007: SAFETY CIRCUIT FAULT
Axis and main drive contactor with machine switched off not disabled. Contactor got stuck or contact error. E4.7 was not active during switch-on.

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 or cabling is defective.
A running program will be stopped, the auxiliary drives will be switched off, the reference position will be lost.
Check fuses or contact EMCO service.

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

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

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

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

6019: VICE TIME EXCEED
The electric vice has not reached a stop position within 30 seconds.
The control or the clamping device board are defective, the vice is stuck. Adjust the proximity switches of the stop position.

6020: VICE FAILURE
When the electric vice is closed, the signal "clamping device clamped" of the clamping device board has failed.
The control, the clamping device board or the wiring are defective.

6022: CLAMPING DEVICE BOARD DEFECTIVE

The signal "clamping device clamped" is constantly released, although no command has been given.
Replace the board.

6024: MACHINE DOOR OPEN

The door was opened while a machine movement.
The program will be aborted.

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.

6040: TOOL TURRET INDEX FAILURE

After WZW procedure drum pressed down by Z-axis. Spindle position wrong or mechanical defect. E4.3=0 in lower state

6041: TOOL CHANGE TIMEOUT

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

6043-6046: TOOL DISK POSITION FAULT

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

6047: TOOL DISK UNLOCKED

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

6048: DIVIDING TIME EXCEEDED

Dividing head stucks, insufficient pressured air supply, hardware defective.
Check for collision, check pressured air supply or contact EMCO service.

6049: INTERLOCKING TIME EXCEEDED

see alarm 6048

6050: M25 AT RUNNING MAIN SPINDLE

Cause: Programming mistake in NC program.
A running program will be aborted.
The auxilliary drives will be switched off.
Remedy: Correct NC program

6064: DOOR AUTOMATIC NOT READY

Cause: pressure failure automatic door
          automatic door stucks mechanically
          limit switch for open end position defective
          security print circuits defect
          cabling defective
          fuses defective
A running program will be aborted.
The auxilliary drives will be switched off.
Remedy: service automatic door

6069: CLAMPING FOR TANI NOT OPEN

When opening the clamping pressure switch does not fall within 400ms. Pressure switch defective or mechanical problem. E22.3

6070: PRESSURE SWITCH FOR TANI MISSING

When closing the clamping pressure switch does not respond. No compressed air or mechanical problem. E22.3

6071: DIVIDING DEVICE NOT READY

Servo Ready Signal from frequency converter missing. Excess temperature drive TANI or frequency converter not ready for operation.

EMCO

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

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

6074: DIVIDING TIME EXCEEDED
Cause: dividing device stucks mechanically
locking switch defective
cabling defective
fuses defective
insufficient compressed-air supply.
A running program will be aborted.
The auxilliary drives will be switched off.
Remedy: Check for collision, check the compressed-air supply or contact the EMCO service.

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

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

7001: NO M6 PROGRAMMED
For an automatic tool change you also have to program a M6 after the T word.

7007: FEED STOP!
The axes have been stopped by the robotics interface (robotics entry FEEDHOLD).

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

7017: REFERENCE MACHINE
Approach the reference point.
When the reference point is not active, manual movements are possible only with key switch at position "setting operation".

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

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

7021: INITIALIZE TOOL TURRET
The tool turret operating was interrupted.
No traversing operation is possible.
Press tool turret key in JOG operation. Message occurs after alarm 6040.

7022: INITIALIZE TOOL TURRET !
see 7021

7023: WAITING TIME MAIN DRIVE!
The LENZE frequency converter has to be separated from the mains supply for at least 20 seconds before you are allowed to switch it on again. This message will appear when the door is quickly openend/ closed (under 20 seconds).

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

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

EMCO

7040: MACHINE DOOR OPEN
The main drive can not be switched on and NC-Start can not 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.

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

7051: DIVIDING HEAD NOT LOCKED!
Either the dividing head is in an undefined position after the machine has been switched on, or the locking signal after a dividing process is missing.
Initiate the dividing process, check, respectively adjust the proximity switch for locking.

7054: VICE OPEN
Cause: the workpiece is not clamped
When switching on the main spindle with M3/M4 alarm 6072 (vice not ready) will be released.
Remedy: Clamp

7055: OPEN TOOL CLAMPING SYSTEM
A tool is clamped in the main spindle and the control does not recognize the corresponding T number.
Eject the tool from the main spindle when the door is open by means of the PC keys "Strg" and " 1 ".

7056: SETTING DATA INCORRECT
An invalid tool number is stored in the setting data.
Delete the setting data in the machine directory xxxxx.pls.

7057: TOOLHOLDER OCCUPIED
The clamped tool cannot be positioned in the tool turret since the position is occupied.
Eject the tool from the main spindle when the door is open by means of the PC keys "Strg" and " 1 ".

7058: RETRACTING THE AXES
The position of the tool turret arm cannot be clearly defined during the tool change.
Open the machine door, push the tool turret magazine backwards to the stop. Move the milling head in the JOG mode upwards to the Z reference switch and then traverse the reference point.

7270: OFFSET COMPENSATION ACTIVE !
Only with PC-MILL 105
Offset compensation activated by the following operation sequence.
- Reference point not active
- Machine in reference mode
- Key switch in manual operation
- Press STRG (or CTRL) and simultaneously 4
This must be carried out if prior to the tool change procedure spindle positioning is not completed (tolerance window too large)

7271: COMPENSATION FINISHED,DATA SAVED !
see 7270

H 5

**PC TURN 50 / 55 / 105 / 120 / 125 / 155
Concept TURN 55 / 105 / 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
The SPS-CAN board is not identified by the control.
Check the interface cable and the power supply of the CAN board.

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.

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.

6015: NO DRIVEN TOOL SPINDLE SPEED
see 6014.

6016: AUTOMATIC TOOL TURRET SIGNAL COUPLED MISSING

6017: AUTOMATIC TOOL TURRET SIGNAL UNCOUPLED MISSING
In the tool turret that can be coupled, the position of the coupling and uncoupling magnet is monitored by means of two proximity switches. It has to be made sure that the coupling is in the rear stop position so that the tool turret can get to the next tool position. Equally, during operation with driven tools the coupling has to be safe in the front stop position.
Check and adjust the cables, the magnet and the stop position proximity switches.

6021: COLLET TIME OUT
During closing of the clamping device the pressure switch has not reacted within one second.

6022: CLAMPING DEVICE BOARD DEFECTIVE
The signal "clamping device clamped" is constantly released, even though no command has been given. Replace the board.

6023: COLLET PRESSURE MONITORING
The pressure switch turns off when the clamping device is closed (compressed air failure for more than 500ms).

EMCO

6024: MACHINE DOOR OPEN
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.

6029: TAILSTOCK QUILL TIME EXCEED
The tailstock quill does not reach a final position within 10 seconds.
Adjust the control and the stop position proximity switches, or the tailstock quill is stuck.

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

6031: QUILL FAILURE

6032: TOOL CHANGE TIMEOUT
see alarm 6041.

6033: TOOL TURRET SYNC ERROR
Hardware defective.
Contact EMCO service.

6037: CHUCK TIMEOUT
The pressure switch does not react within one second when the clamping device is closed.

6039: CHUCK PRESSURE FAILURE
The pressure switch turns off when the clamping device is closed (compressed air failure for more than 500ms).

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.

6042: TOOL TURRET OVERHEAT
Tool turret motor too hot.
With the tool turret a max. of 14 swivel procedures a minute may be carried out.

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.

6045: TOOL TURRET SYNC MISSING
Hardware defective.
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.

EMCO

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.

6059: C-AXIS SWING IN TIMEOUT
C-axis does not swivel in within 4 seconds.
Reason: not sufficient air pressure, and/or mechanics stuck.

6060: C-AXIS INDEX FAILURE
When swivelling in the C-axis the limit switch does not respond.
Check pneumatics, mechanics and limit switch.

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.

6065: LOADER MAGAZINE FAILURE
Loader not ready.
Check if the loader is switched on, correctly connected and ready for operation and/or disable loader (WinConfig).

6066: CLAMPING DEVICE FAILURE
No compressed air at the clamping device
Check pneumatics and position of the clamping device proximity detectors.

6067: NO COMPRESSED AIR
Turn the compressed air on, check the setting of the pressure switch.

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.

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.

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.

7019: PNEUMATIC LUBRICATION MONITORING!
Refill pneumatic oil

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.

7022: COLLECTION DEVICE MONITORING
Time exceed of the swivelling movement.
Check the pneumatics, respectively whether the mechanical system is jammed (possibly a workpiece is jammed).

7023: ADJUST PRESSURE SWITCH !
During opening and closing of the clamping device the pressure switch has to turn off and on once.
Adjust the pressure switch. This alarm does not exist any more for versions starting with PLC 3.10.

7024: ADJUST CLAMPING DEVICE PROXIMITY SWITCH !
When the clamping device is open and the position stop control is active, the respective proximity switch has to feed back that the clamping device is "Open".
Check and adjust the clamping device proximity switch, check the cables.

EMCO

7025 WAITING TIME MAIN DRIVE !
The LENZE frequency converter has to be separated from the mains supply for at least 20 seconds before you are allowed to switch it on again. This message will appear when the door is quickly openend/ closed (under 20 seconds).

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 can not be activated (except special operation mode)
Close the machine to run a program.

7041: GEARBOX COVER OPEN
The main spindle cannot be switched on and NC start cannot be activated.
Close the gearbox cover in order to start a CNC 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.

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.

7054: NO PART CLAMPED
No part clamped, switch-on of the spindle is locked.

7055: CLAMPING DEVICE OPEN
This message indicates that the clamping device is not in clamping state. It disappears as soon as a part is clamped.

emco

H 10

emco

AC95 / ACC ALARMS

Axis Controller Alarms

8000 Fatal Error AC

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

8101 Fatal init error AC

see 8101.

8102 Fatal init error AC
see 8101.

8103 Fatal init error AC
see 8101.

8104 Fatal system error AC
see 8101.

8105 Fatal init error AC
see 8101.

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

8107 PC-COM card not working
see 8106.

8108 Fatal error on PC-COM card
see 8106.

8109 Fatal error on PC-COM card
see 8106.

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

8111 Wrong configuration of PC-COM
see 8110.

8113 Invalid data (pccom.hex)
see 8110.

8114 Programming error on PC-COM
see 8110.

8115 PC-COM packet acknowledge missing
see 8110.

8116 PC-COM startup error
see 8110.

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

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

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

8120 PC interrupt no. unmaskable
see 8119

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

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

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

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

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

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

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

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

8129 Invalid MSD data - axisconfig.
see 8128.

8130 Internal init error AC
see 8128.

8130 Internal init error AC
see 8128.

8132 Axis accessed by multiple channels
see 8128.

EMCO

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

8161 X-Axis synchronization lost
Step loss of the step motor. Causes:
- Axis mechanically blocked
- Axis belt defective
- Distance of proximity detector too large (>0,3mm)
  or proximity detector defective
- Step motor defective

8162 Y-Axis synchronization lost
see 8161

8163 Z-Axis synchronization lost
see 8161

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
Remedy: Stop the program with NC stop or with Reset. Traverse the axis.

8174 INC not allowed
Cause: At the moment the axis is in motion.
Remedy: Wait until the axis stops and then traverse the axis.

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.

EMCO

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.

8183 Gear too high
Cause: The selected gear step is not allowed at the machine.

8184 Invalid interpolaton command

8185 Forbidden MSD data change
see 8175.

8186 MSD file could not be opened
see 8175.

8187 PLC program error
see 8175.

8188 Gear command invalid
see 8175.

8189 Invalid channel assignement
see 8175.

8190 Invalid channel within message
see 8175.

8191 Invalid jog feed unit
Cause: The machine does not support the rotation feed in the JOG operating mode.
Remedy: Order a software update from EMCO.

8192 Invalid axis in command
see 8175.

8193 Fatal PLC error
see 8175.

8194 Thread without length
Cause: The programmed target coordinates are identical to the starting coordinates.
Remedy: Correct the target coordinates.

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.

8212 Rotation axis not allowed
see 8199.

8213 Circle and rotation axis can't be interpolated

8214 Thread and rotation axis cant't be interpolated

8215 Invalid state
see 8199.

8216 No rotation axis for rotation axis switch
see 8199.

8217 Axis type not valid!
Cause: Switching during the rotary axis operating mode when the spindle is running.
Remedy: Stop the spindle and switch over to the rotary axis operating mode.

8218 Referencing round axis without selected round axis!
see 8199.

8219 Thread not allowed without spindle encoder!
Cause: Thread cutting, respectively tapping is only possible with spindles with encoders.

8220 Buffer length exceeded in PC send message!
see 8199.

8221 Spindle release although axis is no spindle!
see 8199.

8222 New master spindle is not valid
Cause: The indicated master spindle is not valid when switching over to the master spindle.
Remedy: Correct the spindle number.

8224 Invalid stop mode
see 8199.

EMCO

8225 Invalid parameter for BC_MOVE_TO_IO!
Cause: The machine is not configured for touch probes. A traversing movement with rotary axis is not allowed during touch probe operating mode.
Remedy: Remove the rotary axis movement from the traversing movement.

8226 Rotary axis switch not valid (MSD data)!
Cause: The indicated spindle does not have a rotary axis.

8228 Rotary axis switch not allowed while axis move!
Cause: The rotary axis has moved during switching over to the spindle operating mode.
Remedy: Stop the rotary axis before switching.

8229 Spindle on not allowed while rotary axis is active!

8230 Program start not allowed due to active spindle rotation axis!

8231 Axis configuration (MSD) for TRANSMIT not valid!
Cause: Transmit is not possible at this machine.

8232 Axis configuration (MSD) for TRACYL not valid!
Cause: Tracyl is not possible at this machine.

8233 Axis not available while TRANSMIT/ TRACYL is active!
Cause: Programming of the rotary axis is not allowed during Transmit/ Tracyl.

8234 Axis control grant removed by PLC while axis interpolates!
Cause: Internal error
Remedy: Delete error with reset and inform EMCO.

8235 Interpolation invalid while axis control grant is off by PLC!
see 8234.

8236 TRANSMIT/TRACYL activated while axis or spindle moves!
see 8234.

8237 Motion through pole in TRANSMIT!
Cause: It is not allowed to move through the coordinates X0 Y0 inTransmit.
Remedy: Alter the traversing movement.

8238 Speed limit in TRANSMIT exceeded!
Cause: The traversing movement gets too close to the coordinates X0 Y0. In order to observe the programmed feed rate, the maximum speed of the rotary axis would have to be exceeded.
Remedy: Reduce the feed rate. Set the value of the C-axis feed limitation in WinConfig, machine data settings / general machine data/ to 0.2. Thus, the feed rate will be automatically reduced near the coordinates X0 Y0.

8239 DAU exceeded 10V limit!
Cause: Internal error
Remedy: Start the software again or install it anew. Report the error to EMCO.

8240 Function not valid during active transformation (TRANSMIT/TRACYL)!
Cause: The Jog and INC operating mode are not possible during Transmit in X/C and during Tracyl in the rotary axis.

8241 TRANSMIT not enabled (MSD)!
Cause: Transmit is not possible at this machine.

8242 TRACYL not enabled (MSD)!
Cause: Tracyl is not possible at this machine.

8243 Round axis invalid during active transformation!
Cause: It is not allowed to program the rotary axis during Transmit/Tracyl.

8245 TRACYL radius = 0!
Cause: When selecting Tracyl, a radius of 0 was used.
Remedy: Correct the radius.

8246 Offset alignment not valid for this state!
see 8239.

8247 Offset alignment: MSD file write protected!

8248 Cyclic supervision failed!
Cause: The communication with the machine keyboard is interrupted.
Remedy: Start the software again or install it anew. Report the error to EMCO.

8249 Axis motion check alarm!
see 8239

8250 Spindle must be rotation axis !
see 8239

8251 Lead for G331/G332 missing !
Cause: The threading pitch is missing or the starting coordinates are identical to the target coordinates.
Remedy: Program the threading pitch.
Correct the target coordinates.

8252 Multiple or no linear axis programmed for G331/G332 !
Remedy: Program exactly one linear axis.

8253 Speed value for G331/G332 and G96 missing !
Cause: No cutting speed has been programmed.
Remedy: Program the cutting speed.

8254 Value for thread starting point offset not valid!
Cause: The thread starting point offset is not within the range of 0 to 360°.
Remedy: Correct the thread starting point offset.

EMCO

8255 Reference point not in valid software limits!
Cause: The reference point has been defined outside the software limit switches.
Remedy: Correct the reference points in WinConfig.

8256 Spindle speed too low while executing G331/G332!
Cause: During tapping the spindle speed has decreased. Perhaps the incorrect threading pitch was used or the core drilling is not correct.
Remedy: Correct the threading pitch. Adapt the diameter to the core drilling.

8257 Real Time Module not active or PCI card not found!
Cause: ACC could not be started correctly or the PCI card in the ACC was not recognized.
Remedy: Report the error to EMCO.

8258 Error allocating Linux data!
see 8239.

8259 Current thread in sequence not valid!
Cause: One block of a thread in sequence has been programmed without thread G33.
Remedy: Correct the program.

8261 Missing thread in sequence !
Cause: A successive thread has not been programmed for a thread in sequence, the number has to be in accordance with the SETTHREADCOUNT () that has been defined before.
Remedy: Correct the number of threads in the thread in sequence and add a thread.

8262 Reference marks are not close enough !
Cause: The settings of the linear scale have been changed or the linear scale is defective.
Remedy: Correct the settings. Contact EMCO.

8263 Reference marks are too close together!
see 8262.

22000 Gear change not allowed
Cause: Gear step change when the spindle is active.
Remedy: Stop the spindle and carry out a gear step change.

22270 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

H 16

I: Control Alarms

Control Alarms 10000 - 59999

These alarms will be triggered by the control.
These are the same alarms as they would appear on
the original SIEMENS control.

10208	**Channel %1 Continue program with NC Start**
Explanation:	%1 = Channel number
	After block search with calculation, the control is in the desired state.
	The program can now be started with NC Start or the state can be changed for the time being with overstore/jog.
Reaction:	Alarm display. NC Stop when alarm.
Remedy:	Clear alarm with NC Start and continue processing.

10209	**Channel %1 internal NC Stop after block search**
Explanation:	%1 = Channel number
	Internal alarm which serves for releasing NC Stop via the alarm reaction. The alarm is output if $MN_SEARCH_RUN_MODE ==1 and the last action block is entered after block search in the main run. The alarm 10208 is activated depending on the VDI signal PLC -> NCK channel DBB1.6.
Reaction:	NC Stop when alarm.
Remedy:	Clear alarm with NC Start and continue processing.

10620	**Channel %1 block %3 axis %2 at software limit switch %4**
Explanation:	%1 = Channel number
	%2 = Axis name, spindle number
	%3 = Block number, label
	%4 = String
	During the traversing motion, it is recognized that the software limit switch would be overtraveled in the displayed direction. It was not yet possible to detect in the block preparation that the traversing range would be exceeded: either there has been a motion overlay by the **handwheel** or a **coordinate transformation** is active.
Reaction:	Alarm display. Interface signals are set. NC Start disable.
Remedy:	Depending on the reason for this alarm being triggered, the following remedial measures should be undertaken:
	1. **Handwheel override:** Cancel the motion overlay and avoid this or keep it smaller when the program is repeated.
	2. **Transformation:** Check the preset/programmed zero offsets (current frame). If the values are correct, the tool holder (fixture) must be moved in order to avoid triggering the same alarm when the program is repeated, which would again cause the program to be aborted.
	Clear alarm with RESET key. Restart part program.

10630	**Channel %1 block %2 axis %3 at working area limit %4**
Explanation:	%1 = Channel number
	%2 = Block number, label
	%3 = Axis, spindle number
	%4 = String (+ or -)
	The specified axis violates the working area limitation. This is recognized only in the main run because either the minimum axis values could not be measured before transformation or because there is a motion overlay.
Reaction:	Alarm display. Interface signals are set. NC Stop when alarm at the block end. NC Start disable.
Remedy:	Program other motion or do not perform overlaid motion.
	Clear alarm with RESET key. Restart part program.

10720	**Channel %1 block %3 axis %2 software limit switch %4**
	%1 = Channel number
	%2 = Axis name, spindle number
	%3 = Block number, label
	%4 = String (+ or -)
	For the axis, the programmed path violates the currently valid software limit switch. (The 2nd software limit switch becomes active with the interface signal "2nd software limit switch plus/minus" in DB 31 - 48, DBX 12.2 and 12.3). The alarm is activated when preparing the part program block.
Reaction:	Alarm display. Interface signals are set. Correction block is reorganized. NC Start disable.
Remedy:	Check the **position** of the axis as specified in the part program.
	Machine data:
	36100 **POS_LIMIT_MINUS**/36120 **POS_LIMIT_MINUS2** and 36110 **POS_LIMIT_PLUS**/36130 **POS_LIMIT_PLUS2** must be checked for the software limit switches.
	Check the axis-specific **interface signals:** „Check 2nd software limit switch plus/minus" (DB 31 - 48, DBX 12.2 and 12.3) to see whether the 2nd software limit switch is selected.
	Check currently active **zero offsets** via the current frame.
	Clear alarm with NC Start and continue program.

emco

10730   **Channel %1 block %3 axis %2 working area limitation %4**
Explanation: %1 = Channel number
       %2 = Axis name, spindle number
       %3 = Block number, label
       %4 = String (+ or -)
       This alarm is generated if it is determined during block preparation that the programmed path of the axis will result in exceeding the working area limitation.
Reaction: Alarm display. Interface signals are set. Correction block is reorganized. NC Start disable.
Remedy   a) Check **NC program** for correct positional data
       b) Check **zero offsets** (current frame)
       c) Correct **working area limitation** via G25, or
       d) Correct **working area limitation** via setting data, or
       e) Deactivate **working area limitation** via setting data: 43410 **WORKAREA_MINUS_ENABLE**=FALSE
       Clear alarm with NC Start and continue program.

10740   **Channel %1 block %2 too many empty blocks in WAB programming**
Explanation: %1 = Channel number
       %2 = Block number, label
       It is not allowed to program more blocks than specified by machine data MC_WAB_MAXNUM_DUMMY_BLOCKS between the WAB block and the block determining the approach and retraction tangent
Reaction: Alarm display. Interface signals are set. Correction block is reorganized. NC Start disable. NC Start when alarm at the block end.
Remedy: Modify part program
       Clear alarm with NC Start and continue program.

10741   **Channel %1 block %2 direction reversal with WAB infeed motion**
Explanation: %1 = Channel number
       %2 = Block number, label
       A safety distance has been programmed which is located vertically to the machining plane and not located between the start and end point of the WAB contour.
Reaction: Alarm display. Interface signals are set. Correction block is reorganized. NC Start disable. NC Start when alarm at the block end.
Remedy: Modify part program
       Clear alarm with NC Start and continue program.

10742   **Channel %1 block %2 WAB distance invalid or not programmed**
Explanation: %1 = Channel number
       %2 = Block number, label
Possible causes: • In a WAB block, the parameter DISR has not been stated or its value is less than or equal to 0.
       • During approach or retraction with circle and active tool radius, the radius of the internally generated WAB contour is negative. The internally generated WAB contour is a circle with such a radius so that when it is offset with the current offset radius (sum of tool radius and offset value OFFN) the tool center point path with the programmed radius DISR result from this.
Reaction: Alarm display. Interface signals are set. Correction block is reorganized. NC Start disable. NC Start when alarm at the block end.
Remedy: Modify part program
       Clear alarm with NC Start and continue program.

10743   **Channel %1 block %2 WAB programmed several times**
Explanation: %1 = Channel number
       %2 = Block number, label
       An attempt has been made to activate a WAB motion before a WAB motion activated previously was terminated.
Reaction: Alarm display. Interface signals are set. Correction block is reorganized. NC Start disable. NC Start when alarm at the block end.
Remedy: Modify part program
       Clear alarm with NC Start and continue program.

10744   **Channel %1 block %2 no valid WAB direction defined**
Explanation: %1 = Channel number
       %2 = Block number, label
       The tangent direction for smooth approach or retraction is not defined.
Possible causes: • In the program, no block with travel information follows the approach block
       • Before a retraction block, no block with travel information has been programmed in a program.
       • The tangent to be used for WAB motion is vertical to the current machining plane.
Reaction: Alarm display. Interface signals are set. Correction block is reorganized. NC Start disable. NC Start when alarm at the block end.
Remedy: Modify part program
       Clear alarm with NC Start and continue program.

10745   **Channel %1 block %2 WAB end positioning not clear**
Explanation: %1 = Channel number
       %2 = Block number, label
       In the WAB block and in the following block, the position has been programmed vertically to the machining direction. In the WAB block, no position has been indicated in the machining plane.
Reaction: Alarm display. Interface signals are set. Correction block is reorganized. NC Start disable. NC Start when alarm at the block end.
Remedy: Modify part program

Either remove the position data for the infeed axis from the WAB block or from the following block, or program also a position in the machining plane in the WAB block.
Clear alarm with NC Start and continue program.

10746   **Channel %1 block %2 block search stop for WAB**
Explanation: %1 = Channel number
%2 = Block number, label
A block search stop has been inserted between a WAB approach block and the following block defining the tangent direction or between a WAB approach block and the following block defining the end position.
Reaction: Alarm display. Interface signals are set. Correction block is reorganized. NC Start disable. NC Start when alarm at the block end.
Remedy: Modify part program.
Clear alarm with NC Start and continue program.

10747   **Channel %1 block %2 retraction direction not defined for WAB**
Explanation: %1 = Channel number
%2 = Block number, label
In a WAB retraction block with quarter circle or semi-circle (G248 or G348), the end point in the machining plane was not programmed, and either G143 or G140 without tool radius compensation is active.
Reaction: Alarm display. Interface signals are set. Correction block is reorganized. NC Start disable. NC Start when alarm at the block end.
Remedy: Modify part program.
The following changes are possible:
• Indicate end point in the machining plane in the WAB block.
• Activate tool radius compensation (effective for G140 only, not for G143)
• State retraction side explicitly with G141 or G142.
• Perform retraction with a straight line instead of a circle.
Clear alarm with NC Start and continue program.

10750   **Channel %1 block %2 tool radius compensation activated without tool no.**
Explanation: %1 = Channel number
%2 = Block number, label
A tool T... must be selected so that the control can make allowance for the associated compensation values. A correction data block (D1) containing the correction values (parameter P1 -P25) is automatically assigned to each tool (T number). Up to 9 correction data blocks can be assigned to a tool by specifying the required data block with the D number (D1 - D9). The cutter radius compensation (CRC) is allowed for if function G41 or G42 is programmed. The correction values are contained in parameter P6 (geometry value) and P15 (wear value) of the active correction data block D x .
Reaction: Alarm display. Interface signals are set. Correction block is reorganized NC Start disable. NC Stop when alarm at block end.
Remedy: Before calling the CRC with G41/G42, program a tool number under the address T... .
Clear alarm with NC Start and continue program.

10751   **Channel %1 block %2 danger of collision due to tool radius compensation**
Explanation: %1 = Channel number
%2 = Block number, label
The „Bottleneck detection" (calculation of intersection for the following compensated traversing blocks) has not been able to calculate a point of intersection for the reviewed number of traversing blocks. It is therefore possible that one of the equidistant paths violates the workpiece contour.
Reaction: Alarm display. Interface signals are set. Correction block is reorganized. NC Start disable. NC Start when alarm at the block end.
Remedy: Check the part program and modify the programming if possible such that inside corners with smaller paths than the correction value are avoided. (Outside corners are not critical because the equidistants are lengthened or intermediate blocks are inserted so that there is always a point of intersection). Increase the number of reviewed traversing blocks via machine data 20240 **CUTCOM_MAXNUM_CHECK_BLOCKS** (default: 3), resulting in an increase in the extent of calculation and therefore also the block cycle time.
Clear alarm with NC Start and continue program.

10753   **Channel %1 block %2: activate tool radius compensation in linear block**
Explanation: %1 = Channel number
%2 = Block number, label
Selection of cutter radius compensation with G41/G42 may only be performed in blocks where the G function G00 (rapid traverse) or G01 (feed) is active. In the block with G41/G42, at least one axis in the plane G17 to G19 must be written. It is always advisable to write both axes because as a rule, both axes are traversed when selecting the compensation.
Reaction: Alarm display. Interface signals are set. Correction block is reorganized. NC Start disable. NC Stop when alarm at block end.
Remedy: Correct the NC program and put the compensation selection in a block with linear interpolation.
Clear alarm with NC Start and continue program.

10754   **Channel %1 block %2: deactivate tool radius compensation in linear block only**
Explanation: %1 = Channel number
%2 = Block number, label
Deselection of cutter radius compensation with G40 can only be performed in blocks where the G function G00 (rapid traverse) or G01 (feed) is active. In the block with G40, at least one axis in the plane G17 to G19 must be written.

EMCO

It is always advisable to write both axes because as a rule, both axes are traversed when deselecting the compensation.

Reaction: Alarm display. Interface signals are set. Correction block is reorganized. NC Start disable. NC Stop when alarm at block end.

Remedy: Correct the NC program and put the compensation selection in a block with linear interpolation.
Clear alarm with NC Start and continue program.

10755
Explanation: **Channel %1 block %2: do not activate tool radius compensation via KONT at the current starting point**
%1 = Channel number
%2 = Block number, label
When activating the cutter radius compensation with **KONT** the starting point of the approach block is within the compensation circle and therefore already violates the contour.
If the cutter radius compensation is selected with G41/G42, the approach behaviour (NORM or KONT) determines the compensation movement if the momentary actual position is **behind** the contour. With KONT, a circle is drawn with the cutter radius around the programmed initial point (= end point of the approach block). The tangent that passes through the current actual position and does not violate the contour is the approach movement. If the start point is within the compensation circle around the target point, no tangent passes through this point.

Reaction: Alarm display. Interface signals are set. Correction block is reorganized. NC Start disable. NC Stop when alarm at block end.

Remedy: Place selection of the CLC such that the starting point of the approach movements comes to rest outside of the correction circle around the target point (programmed traversing movements > compensation radius). The following possibilities are available:
• Selection in the previous block
• Insert intermediate block
• Select approach behaviour NORM
Clear alarm with NC Start and continue program.

10756
Explanation: **Channel %1 block %2: do not deactivate tool radius compensation via KONT at the programmed end point**
%1 = Channel number
%2 = Block number, label
On deselection of the cutter radius compensation, the programmed end point is within the compensation circle. If this point were in fact to be approached without compensation, there would be a contour violation. If the cutter radius compensation is deselected via G40, the approach behaviour (NORM or KONT) determines the compensation movement if the programmed end point is **behind** the contour. With KONT, a circle is drawn with the cutter radius about the last point at which the compensation is still active. The tangent passing through the programmed end position and not violating the contour is the retraction movement. If the start point is within the compensation circle around the target point, no tangent passes through this point.

Reaction: Alarm display. Interface signals are set. Correction block is reorganized. NC Start disable. NC Stop when alarm at block end.

Remedy: Place deselection of the CLC such that the programmed end point comes to rest outside of the compensation circle about the last active compensation point.
The following possibilities are available:
• Deselection in the next block
• Insert intermediate block
• Select retract behaviour NORM
Clear alarm with NC Start and continue program.

10757
Explanation: **Channel %1 block %2: do not change the compensation plane while tool radius compensation is active**
%1 = Channel number
%2 = Block number, label
In order to change the compensation plane (G17, G18 or G19) it is first necessary to deselect the cutter radius compensation with G40.

Reaction: Alarm display. Interface signals are set. Correction block is reorganized. NC Start disable. NC Stop when alarm at block end.

Remedy: Insert an intermediate block with compensation deselection in the part program. After plane change, the cutter radius compensation must be selected in an approach block with linear interpolation.
Clear alarm with NC Start and continue program.

10758
Explanation: **Channel %1 block %2 curvature radius with variable compensation value too small**
%1 = Channel number
%2 = Block number, label
The current cutter radius compensation (the cutter used) is too large for the programmed path radius. In a block with variable tool radius compensation, a compensation must be possible either anywhere or nowhere on the contour with the smallest **and** the largest compensation value from the programmed range. There must be no point on the contour in which the curvature radius is within the variable compensation range. If the compensation value varies its sign within a block, both sides of the contour are checked, otherwise only the compensation side.

Reaction: Alarm display. Interface signals are set. Correction block is reorganized. NC Start disable. NC Stop when alarm at block end.

Remedy: Use smaller cutters or allow for a part of the cutter radius at the time of contour programming.
Clear alarm with NC Start and continue program.

10760
Explanation: **Channel %1 block %2 helical axis is not parallel to tool orientation**
%1 = Channel number
%2 = Block number, label

With active tool radius compensation a helix is only permissible if the helix axis is parallel to the tool, i.e. the circle plane and the compensation plane must be identical.

Reaction:	Alarm display. Interface signals are set. Correction block is reorganized. NC Start disable. NC Stop when alarm at block end.
Remedy:	Orient helix axis perpendicular to the machining plane. Clear alarm with NC Start and continue program.

10762   **Channel %1 block %2 too many empty blocks between two traversing blocks with active tool radius compensation**

Explanation:	%1 = Channel number %2 = Block number, label The maximum permissible number of empty blocks are limited by a machine data.
Reaction:	Alarm display. Interface signals are set. Correction block is reorganized. NC Start disable. NC Stop when alarm at block end.
Remedy:	1. Modify part program: 2. Modify machine data 3. Check whether SBL2 is activated. With SBL2, a block is generated from each part program line which can lead to exceeding the maximum permissible number of empty blocks between two traversing blocks. Clear alarm with NC Start and continue program.

10763   **Channel %1 block %2 path component of the block in the compensation plane becomes zero.**

Explanation:	%1 = Channel number %2 = Block number, label Due to the collision monitoring with active tool radius compensation, the path component of the block in the compensation plane becomes zero. If the original block contains no motion information perpendicular to the compensation plane, it means that this block is excluded.
Reaction:	Alarm display.
Remedy:	• The behaviour is correct at narrow locations that cannot be machined with the active tool. • Modify the part program if necessary • Use tool with smaller radius if necessary • Program CDOF. Clear alarm with the Cancel key. No further operator action necessary.

10764   **Channel %1 block %2 discontinuous path with active tool radius compensation**

Explanation:	%1 = Channel number %2 = Block number, label This alarm occurs when, with active tool radius compensation, the starting point used for calculating the compensation is not identical to the end point of the preceding block. This situation can occur, for example, when a geometry axis is traversed between two positions as positioning axis or when, with an active kinematic transformation (e.g. 5-axis transformation) the tool length compensation is altered.
Reaction:	Alarm display. Interface signals are set. Correction block is reorganized. NC Start disable. NC Stop when alarm at block end.
Remedy:	Modify part program. Clear alarm with NC Start and continue program.

10770   **Channel %1 block %2 change of corner type due to change of orientation with active tool radius compensation**

Explanation:	%1 = Channel number %2 = Block number, label The type of a corner (inside or outside corner) depends not only on the programmed path but also on the tool orientation. For this purpose, the programmed path is projected in the plane perpendicularly to the actual tool orientation and the corner type is determined there. If a change in orientation is programmed (in one or several blocks) between two traversing blocks, resulting in the type of corner at the end of the first traversing block being different from that at the start point of the second block, the above error message is issued.
Reaction:	Alarm display. Interface signals are set. Correction block is reorganized. NC Start disable. NC Stop when alarm at block end.
Remedy:	Modify part program. Clear alarm with NC Start and continue program.

10774   **Channel %1 illegal tool dimensions with face cutting in block %2**

Explanation:	%1 = Channel number %2 = Block number, label This alarm occurs when illegal tool dimensions are programmed for face milling, e.g. negative tool radius, rounding radius zero or negative for tool types that require a rounding radius, taper angle zero or negative for tapered tools.
Reaction:	Alarm display. Interface signals are set. Correction block is reorganized. NC Start disable. NC Stop when alarm at block end.
Remedy:	Modify part program Clear alarm with NC Start and continue program.

10776   **Channel %1 block %2 axis %3 must be geo axis if cutter compensation is active**

Explanation:	%1 = Channel number %2 = Block number, label %3 = axis name This alarm occurs when an axis that is required for tool radius compensation is not a geometry axis. With CUT2DF, the axis can be a positioning axis perpendicular to the machining plane; with all other types of compensation

(CUT2DF, CUT3DC, CUT3DF, CUT3DFF), all geometry axes must be operated as such.

Reaction:	Alarm display. Interface signals are set. Correction block is reorganized. NC Start disable. NC Stop when alarm at block end.
Remedy:	Modify part program.
	Clear alarm with NC Start and continue program.

10777   **Channel %1 block %2 tool radius compensation: too many blocks with suppression of compensation**

Explanation: %1 = Channel number
%2 = Block number, label
The maximum permissible number of blocks with active compensation suppression with tool radius compensation is limited by the machine data CUTCOM_MAXNUM_SUPPRESS_BLOCKS.

Reaction: Alarm display Interface signals are set Correction block mit Reorganisieren NC Start disable NC Stop when alarm at block end

Remedy: • Modify part program
• Modify machine data
• Check whether SBL2 is activated. With SBL2, a block is generated from each part program line which can lead to exceeding the maximum permissible number of empty blocks between two traversing blocks.
Clear alarm with NC Start and continue program.

10 778   **Channel %1 block %2 preparation stop with active tool radius compensation**

Explanation: %1 = Channel number
%2 = Block number, label
If a preprocessing stop is detected with active tool radius compensation (either programmed by the user or generated internally) and the setting data $SC_STOP_CC_STOPRE is set, then this warning is issued because in this situation machine movements which were not intended by the user can occur (termination of radius compensation and new approach).
To continue machining, activate the CANCEL key and perform a restart.

Reaction: Alarm display. NC Stop when alarm at block end.

Remedy: • Continue machining with CANCEL and Start
• Modify part program
• Set setting data $SC_STOP_CC_STOPRE to FALSE.
Clear alarm with the Cancel key. No further operator action necessary.

10 800   **Channel %1 block %3 axis %2 is not a geometry axis**

Explanation: %1 = Channel number
%2 = Axis name, spindle number
%3 = Block number, label
With an active **transformation** or a **frame with a rotation component** the geometry axes are needed for block preparation. If a geometry axis has previously been traversed as positioning axis, it retains its status of „positioning axis" until it is again programmed as a geometry axis. Because of the POSA motion beyond block boundaries, it is not possible to identify in the preprocessing run whether the axis has already reached its target position when the block is executed. This is, however, an unconditional requirement for calculating the ROT component of the frame or of the transformation.
If geometry axes are used as positioning axes, then:
1. **No rotation** may be specified in the current overall frame.
2. **No transformation** may be selected.

Reaction: Alarm display. Interface signals are set. Correction block is reorganized. NC Start disable.

Remedy: After selecting transformation or frame, reprogram the geometry axis now operating as positioning axis (e.g. with WAITP) in order to revert the status to „geometry axis"..
Clear alarm with NC Start and continue program.

10805   **Channel %1 block %2 repositioning after switch of geoaxes or transformation**

Explanation: %1 = Channel number
%2 = Block number, label
In the asynchronous subroutine the assignment of geometry axes to channel axes was changed or the active transformation modified.

Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.

Remedy: Modify part program
Clear alarm with RESET key. Restart part program.

10810   **Channel %1 block %2 master spindle not defined**

Explanation: %1 = Channel number
%2 = Block number, label
The function „Revolutional feedrate" (with G95 or G96), or „Rigid tapping" (with G331/G332) has been programmed, although no master spindle is defined from which the speed could be derived. For the definition the MD 20090 **SPIND_DEF_MASTER_SPIND** is available for the default or the keyword **SETMS** in the part program, thus allowing each spindle of the channel to be redefined as master spindle.

Reaction: Alarm display. Interface signals are set Correction block is reorganized. NC Start disable.

Remedy: Preset master spindle with MD 20090 **SPIND_DEF_MASTER_SPIND[n]=m** (n ... channel index, m ... spindle no.) or define it in the NC part program with an identifier, before a G function is programmed that requires a spindle. The machine axis to be operated as spindle must be provided with a spindle number in MD 35000 **SPIND_ASSIGN_TO_MACHAX[n]=m** (n ... machine axis index, m ... spindle no.). Furthermore, it must be assigned to a channel (channel axis index 1 or 2) with the MD 20070 **AXCONF_MACHAX_USED[n]=m** (n ... channel axis index, m ... machine axis index)..

EMCO

Clear alarm with NC Start and continue program.

10820   **Channel %1 rotary axis/spindle %2 not defined**

Explanation: %1 = Channel number
%2 = Axis name, spindle number
Revolutional feed has been programmed for contouring and synchronous axes or for an axis/spindle. However, the rotary axis/spindle from which the feed is to be deduced is not available.

Reaction: Alarm display. Interface signals are set Correction block is reorganized. NC Start disable.

Remedy: Correct part program or set the setting data 43300 ASSIGN_FEED_PER_REV_SOURCE correctly.
Clear alarm with NC Start and continue program.

10860   **Channel %1 block %2 feedrate not programmed**

Explanation: %1 = Channel number
%2 = Block number, label
In the displayed block, an interpolation type other than G00 (rapid traverse) is active.
The F value has not been programmed.

Reaction: Alarm display. Interface signals are set Correction block is reorganized. NC Start disable.

Remedy: Program feedrate in accordance with the interpolation type.
G93: The feedrate is specified as a time-reciprocal value under address **F** in [1/min].
G94 and **G97:** The feedrate is programmed under address **F** in [mm/min] or [m/min].
G95: The feedrate is programmed as revolutional feedrate under address **F** in [mm/revolution].
G96: The feedrate is programmed as cutting rate under address **S** in [m/min]. It is derived from the current **spindle speed.**
Clear alarm with NC Start and continue program.

10861   **Channel %1 block %2 velocity of positioning axis %3 is zero**

Explanation: %1 = Channel number
%2 = Block number, label
%3 = axis
No axis velocity has been programmed and the positioning velocity set in the machine data is zero.

Reaction: Alarm display. Interface signals are set Correction block is reorganized. NC Start disable.

Remedy: Enter a different velocity in machine data 32060 **MA_POS_AX_VELO**.
Clear alarm with NC Start and continue program.

10862   **Channel %1 block %2 master spindle is axis of path**

Explanation: %1 = Channel number
%2 = Block number, label
A contour has been programmed that also includes the master spindle as contouring axis.
However, the velocity of the contour is derived from the rotational speed of the master spindle (e.g. G95).

Reaction: Alarm display. Interface signals are set Correction block is reorganized. NC Start disable.

Remedy: Modify the program so that no reference is possible to the program itself.
Clear alarm with NC Start and continue program.

10870   **Channel %1 block %2 facing axis not defined**

Explanation: %1 = Channel number
%2 = Block number, label
When constant cutting speed is activated via the **G96** function, the spindle speed is controlled through the position of the facing axis such that the cutting speed programmed under S [mm/min] is applied at the tool tip. In the channel-specific MD 20100 **DIAMETER_AX_DEF[n,m]=x** (n ... channel index, m ... spindle index, x ... axis name), the name of the facing axis [String] can be specified for each of the 5 spindles used for speed calculation.

$$S[1/min] = \frac{S_{G96}[m/min] \cdot 1000}{D_{face\_axis}[mm] \cdot \pi}$$

Reaction: Alarm display. Interface signals are set Correction block is reorganized. NC Start disable.

Remedy: Enter the name of the facing axis in the channel-specific machine data 20100 **DIAMETER_AX_DEF** for the spindles used.
Clear alarm with NC Start and continue program.

10880   **Channel %1 block %2 too many empty blocks between two traversing blocks when inserting chamfer or radius**

Explanation: %1 = Channel number
%2 = Block number, label
Between 2 blocks containing contour elements and which are to be joined with a chamfer or a radius **(CHF, RND)**, more blocks without contour information have been programmed than provided for in the machine data 20200 **CHFRND_MAXNUM_DUMMY_BLOCKS**.

Reaction: Alarm display. Interface signals are set Correction block is reorganized. NC Start disable.

Remedy: Modify the **part program** in order that the permissible number of dummy blocks is not exceeded or adapt the channel-specific machine data 20200 **CHFRND_MAXNUM_DUMMY_BLOCKS** (dummy blocks with chamfers/radii) to the maximum number of dummy blocks.
Clear alarm with NC Start and continue program.

10882   **Channel %1 block %2: do not activate chamfer or radius without traversing**

Explanation: %1 = Channel number
%2 = Block number, label
No chamfer or radius has been inserted between 2 linear or circle contours (edge breaking) because:

• There is no straight line or circle contour in the plane
• There is a movement outside of the plane
• A plane change has taken place
• The permissible number of dummy blocks without traversing information has been exceeded

Reaction: Alarm display. Interface signals are set Correction block is reorganized. NC Start disable.

Remedy: Correct the part program according to the above error description or change the number of dummy blocks in the channel-specific MD **CHFRND_MAXNUM_DUMMY_BLOCKS** to comply with the maximum number allowed for in the program.
Clear alarm with NC Start and continue program.

10900   **Channel %1 block %2 no S value programmed for constant cutting speed**

Explanation: %1 = Channel number
%2 = Block number, label
If G96 is active, the constant cutting speed under address S is missing

Reaction: Alarm display. Interface signals are set Correction block is reorganized. NC Start disable.

Remedy: Program constant cutting speed under S in [m/min] or deselect the function G96. For example, with G97 the previous feed is retained but the spindle continues to rotate at the momentary speed.
Clear alarm with NC Start and continue program.

10910   **Channel %1 block %2 excessive velocity of one path axis**

Explanation: %1 = Channel number
%2 = Block number, label
With active transformation, an excessive increase in velocity occurs in one or several axes, e.g. because the path passes close by the pole.

Reaction: Alarm display.

Remedy: Divide the NC block into several blocks (e.g. 3) so that the path section with the excess is as small as possible and therefore of short duration. The other blocks are then traversed at the programmed velocity.
Clear alarm with Cancel key. No further operator action necessary.

10911   **Channel %1 block %2 transformation prohibits to traverse the pole.**

Explanation: %1 = Channel number
%2 = Block number, label
The given curve passes through the pole of the transformation.

Reaction: Alarm display. Interface signals are set. NC Start disable.

Remedy: Modify part program.
Clear alarm with RESET key. Restart part program.

10914   **Movement not possible while transformation active - in channel %1 for block %2**

Explanation: %1 = Channel number
%2 = Block number, label
The machine kinematics does not allow the specified motion.

Reaction: If the working area limitation is violated (see machine position), the part program's working area must be changed such that the possible operating range be adhered to (e.g. modified part settings).

Remedy: Clear alarm with the RESET key. Restart part program.

10930   **Channel %1 block %2 interpolation type not allowed in stock removal contour**

Explanation: %1 = Channel number
%2 = Block number, label
The contour of the stock removal cycle contains positioning commands other than G00, G01, G02 or G03. The contour program may contain only such contour elements as built up on these preparatory functions (i.e. no threading blocks, no spline blocks etc.).

Reaction: Alarm display. Interface signals are set. NC Start disable.

Remedy: In the contour subroutine, program only path elements that consist of straight lines and circular arcs.
Clear alarm with RESET key. Restart part program.

10931   **Channel %1 block %2 error in programmed stock removal contour**

Explanation: %1 = Channel number
%2 = Block number, label
In the subroutine for the contour there are the following errors during stock removal:
• Full circle
• Overlapping contour elements
• Wrong start position

Reaction: Alarm display. Interface signals are set. NC Start disable.

Remedy: The errors listed above must be corrected in the subroutine for the stock removal contour.
Clear alarm with RESET key. Restart part program.

10932   **Channel %1 block %2 preparation of contour has been restarted**

Explanation: %1 = Channel number
%2 = Block number, label
After contour segmentation has been started with the keyword CONTPRON, the contour to be prepared is described in the following block (as subroutine and/or main program). Following contour description, the contour segmentation must be ended with the keyword EXECUTE before a new call may occur.

Reaction: Alarm display. Interface signals are set. NC Start disable.

Remedy: Program the keyword EXECUTE for ending the previous conditioning in the part program before again calling up contour segmentation (keyword CONTPRON).
Clear alarm with RESET key. Restart part program.

EMCO

10933   **Channel %1 block %2 contour program contains too few contour blocks**
Explanation: %1 = Channel number
%2 = Block number, label
The subroutine in which the stock removal contour is programmed contains fewer than 3 blocks with movements in both axes in the machining plane.
The stock removal cycle has been aborted.
Reaction: Alarm display. Interface signals are set. NC reagiert innerhalb einer Bearbeitungsstation. NC Start disable.
Remedy: Increase the size of the subroutine with the stock removal contour to include at least 3 NC blocks with movements in both axes of the current machining plane.
Clear alarm with RESET key. Restart part program.

10934   **Channel %1 block %2 array for contour segmentation is set too small**
Explanation: %1 = Channel number
%2 = Block number, label
During contour segmentation (activated with the keyword CONTPRON), the field for the contour table has been detected as too small. For every permissible contour element (circle or straight line) there must be a row in the contour table.
Reaction: NC reacts within a machining station. Alarm display. Interface signals are set. NC Start disable.
Remedy: Base the definition of the field variables of the contour table on the contour elements to be expected. The contour segmentation function divides up some NC blocks into as many as 3 machining cuts.
Example:
N100 DEF TABNAME_1 [30, 11] Field variables for the contour table provide for 30 machining cuts. The number of columns (11) is a fixed quantity.
Clear alarm with RESET key. Restart part program.

12000   **Channel %1 block %2 address %3 programmed repeatedly**
Explanation: %1 = Channel number
%2 = Block number, label
%3 = Source string der Adresse
Most addresses (address types) may only be programmed **once** in an NC block, so that the block information remains unambiguous (e.g. X... T... F... etc. - exception: G and M functions).
Reaction: Alarm display. Interface signals are set Correction block.
Remedy: Press the NC Stop key and select the function „Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block.
• Remove from the NC program addresses that occur more than once (except for those where multiple value assignments are allowed).
• Check whether the address (e.g. the axis name) is specified via a user-defined variable (this may not be easy to see if allocation of the axis name to the variable is performed in the program through computational operations only).
Clear alarm with NC Start and continue processing.

12010   **Channel %1 block %2 address %3 address type programmed too often**
Explanation: %1 = Channel number
%2 = Block number, label
%3 = Source string of the address
For each address type, it is defined internally how often it may occur in a DIN block (for instance, all axes together form **one** address type for which a block limit also applies).
Reaction: Alarm display. Interface signals are set Correction block.
Remedy: Press the NC Stop key and select the function „Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. The program information must be split up over several blocks.
But make sure that the functions are of the non-modal type
Clear alarm with NC Start and continue processing.

12020   **Channel %1 block %2 combination of address modification not allowed**
Explanation: %1 = Channel number
%2 = Block number, label
Valid address types are 'IC', 'AC', 'DC', 'CIC', 'CAC', 'ACN', 'ACP', 'CACN', 'CACP'. Not each of these address modifications can be used for each address type. The Programming Guide specifies which of these can be used for the various address types. If this address modification is applied to address types that are not allowed, then the alarm is generated, e.g.: N10 G02 X50 Y60 **I=DC(20)** J30 F100
interpolation parameters with DC.
Reaction: Alarm display. Interface signals are set Correction block.
Remedy: Press the NC Stop key and select the function „Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block.
Apply non-modal address modifications only for permissible addresses, in accordance with the Programming Guide.
Clear alarm with NC Start and continue processing.

12030   **Channel %1 block %2 invalid arguments or data types in %3**
Explanation: %1 = Channel number
%2 = Block number, label
%3 = Source string
In polynomial interpolation, polynomials must not be greater than the 3rd degree. (Refer to Programming Guide.)
$f(p) = a_0 + a_1 p + a_2 p^2 + a_3 p^3$
The coefficients a_0 (the starting points) are identical to the end points of the preceding block and need not be programmed. In the polynomial block, a **maximum of 3 coefficients per axis** is therefore allowed (a_1, a_2, a_3

emco

).
Reaction: Alarm display. Interface signals are set Correction block.
Remedy: Press the NC Stop key and select the function „Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block.
Clear alarm with NC Start and continue processing.

12040   **Channel %1 block %2 expression %3 is not of data type 'AXIS'**
Explanation: %1 = Channel number
%2 = Block number, label
%3 = Source string in the block
Some keywords demand in their following parameter specification the data to be in variables of the type „AXIS". For example, in the keyword PO the axis identifier must be specified in the parenthesized expression, and it must be defined as a variable of the AXIS type. With the following keywords only parameters of the AXIS type are possible:
AX[..], FA[..], FD[..], FL[..], IP[..], OVRA[..], PO[..], POS[..], POSA[..]
Example:
N5 DEF INT INFEED=Z1 ; incorrect, this does not specify an axis; identifier but the number "26 161"
N5 DEF AXIS INFEED=Z1 ; correct

:   N10 POLY PO[X]=(0.1,0.2,0.3) PO[Y]=(22,33,44) &PO[INFEED]=(1,2,3)
Reaction: Alarm display. Interface signals are set Correction block.
Remedy: Press the NC Stop key and select the function „Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block.
Correct the part program in accordance with the instructions given in the Programming Guide.
Clear alarm with NC Start and continue processing.

12060   **Channel %1 block %2 same G group programmed repeatedly**
Explanation: %1 = Channel number
%2 = Block number, label
The G functions that can be used in the part program are divided into groups that are **syntax defining** or **non-syntax defining**. Only **one G function** may be programmed from each G group. The functions within a group are mutually preclusive. The alarm refers only to the non-syntax defining G functions. If several G functions from these groups are called in one NC block, the **last** of these in a group is active **in each case** (the previous ones are ignored).
G FUNCTIONS:
Syntax defining G functions: 1st to 4th G group
Non-syntax defining G functions: 5th to n G group
Reaction: Alarm display. Interface signals are set. Correction block.
Remedy: Press the NC Stop key and select the function „Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. Remedy is not necessary, but it should be checked whether the G function last programmed really is the one required.
Clear alarm with NC Start and continue processing.

12070   **Channel %1 block %2 too many syntax-defining G functions**
Explanation: %1 = Channel number
%2 = Block number, label
Syntax defining G functions determine the structure of the part program block and the addresses contained in it. **Only one** syntax defining G function may be programmed in each NC block. The G functions in the 1st to 4th G group are syntax defining.
Reaction: Alarm display. Interface signals are set Correction block.
Remedy: Press the NC Stop key and select the function „Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. Analyze NC block and distribute the G functions over several NC blocks.
Clear alarm with NC Start and continue processing.

12080   **Channel %1 block %2 syntax error in text %3**
Explanation: %1 = Channel number
%2 = Block number, label
%3 = Source text area
At the text position shown, the grammar in the block is incorrect. The precise reason for this error cannot be specified in more detail because there are too many possibilities.
Example 1:
N10 IF GOTOF ... ; the condition for the jump is missing!
Example 2:
N10 DEF INT VARI=5
N11 X VARI; the operation is missing for the X and VARI variables
Reaction: Alarm display. Interface signals are set Correction block.
Remedy: Press the NC Stop key and select the function „Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. Analyze the block and correct it in accordance with the syntax rules given in the Programming Guide.
Clear alarm with NC Start and continue processing.

12090   **Channel %1 block %2 unexpected argument %3**
Explanation: %1 = Channel number
%2 = Block number, label

%3 = Disallowed parameters in the text
The programmed function has been predefined; no parameters are allowed in its call. The first unexpected parameter is displayed.
Example:
On calling the predefined subroutine TRAFOF (switching off a transformation) parameters have been transferred (one or more).

Reaction: Alarm display. Interface signals are set Correction block.
Remedy: Press the NC Stop key and select the function „Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block.
Program function without parameter transfer.
Clear alarm with NC Start and continue processing.

12100   **Channel %1 block %2 number of passes %3 not permissible**
Explanation: %1 = Channel number
%2 = Block number, label
%3 = Number of passes
The subroutines called with **MCALL** are modal, i.e. after each block with positional information a routine run is automatically performed **once**. For this reason, programming of the number of passes under address P is not allowed.
The modal call is effective until another MCALL is programmed, either with a new subroutine name or without (delete function).

Reaction: Alarm display. Interface signals are set. Correction block.
Remedy: Press the NC Stop key and select the function „Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block.
Program the subroutine call MCALL without number of passes.
Clear alarm with NC Start and continue processing.

12110   **Channel %1 block %2 syntax cannot be interpreted**
Explanation: %1 = Channel number
%2 = Block number, label
The addresses programmed in the block are not permissible together with the valid syntax defining G function. Example: G1 I10 X20 Y30 F1000. An interpolation parameter must not be programmed in the linear block.
Reaction: Alarm display. Interface signals are set. Correction block.
Remedy: Press the NC Stop key and select the function „Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block.
Check the block structure and correct in accordance with the programming requirements.
Clear alarm with NC Start and continue processing.

12120   **Channel %1 block %2: Write special G function in separate block**
Explanation: %1 = Channel number
%2 = Block number, label
The G function programmed in this block must be alone in the block. No general addresses or synchronous actions may occur in the same block.
These G functions are:
G25, G26 Working area and spindle speed limitation
G110, G111, G112 Pole programming with polar coordinates
G92 Spindle speed limitation with v constant **STARTFIFO, STOPFIFO** Control of preprocessing buffer.
E.g. G4 F1000 M100: no M function allowed in the G4 block.
Reaction: Alarm display. Interface signals are set. Correction block.
Remedy: Program G function by itself in the block.
Clear alarm with NC Start and continue processing.

12140   **Channel %1 block %2 expression %3 not contained in this release**
Explanation: %1 = Channel number
%2 = Block number, label
%3 = Software construct in the source text
In the full configuration of the control functions are possible that are not yet implemented in the current version.
Reaction: Alarm display. Interface signals are set. Correction block.
Remedy: Press the NC Stop key and select the function „Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block.
The displayed function must be removed from the program.
Clear alarm with NC Start and continue processing.

12150   **Channel %1 block %2 operation %3 not compatible with data type**
Explanation: %1 = Channel number
%2 = Block number, label
%3 = String (violating operator)
The data types are not compatible with the required operation (within an arithmetic expression or in a value assignment).
Example 1:
Arithmetic operation
N10 DEF INT OTTO
N11 DEF STRING[17] ANNA
N12 DEF INT MAX
:

N50 MAX = OTTO + ANNA
Example 2:
Value assignment
N10 DEF AXIS BOHR
N11 DEF INT OTTO
:
N50 OTTO = BOHR

Reaction: Alarm display. Interface signals are set. Correction block.
Remedy: Press the NC Stop key and select the function „Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block.
Alter the definition of the variables used such that the required operations can be executed.
Clear alarm with NC Start and continue processing.

12160 **Channel %1 block %2 range of values exceeded**
Explanation: %1 = Channel number
%2 = Block number, label
The programmed constant or the variable exceeds the value range that has previously been established by the definition of data type.
Reaction: Alarm display. Interface signals are set. Correction block.
Remedy: Press the NC Stop key and select the function „Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. Correct value of the constant or adapt data type. If the value for an integer constant is too great, it can be specified as real constant by adding a decimal point.
Example:
R1 = 9 876 543 210 Correct: R1 = 9 876 543 210.
Value range INTEGER: $2^{31} - 1$
Value range REAL:: 2^{-1022} bis 2^{+1023}
Clear alarm with NC Start and continue processing.

12170 **Channel %1 block %2 identifier %3 defined repeatedly**
Explanation: %1 = Channel number
%2 = Block number, label
%3 = Symbol in block
The symbol shown in the error message has already been defined in the active part program. Note that user-defined identifiers may occur more than once if the multiple definition occurs in other (sub)programs i.e. local variables may be redefined with the same name if the program has been exited (subprograms) or has already been concluded. This applies both to user-defined symbols (labels, variables) and to machine data (axes, DIN addresses and G functions).
Reaction: Alarm display. Interface signals are set. Correction block.
Remedy: The symbol already known to data management is displayed. This symbol must be looked for in the definition part of the current program using the program editor. The 1st or 2nd symbol must be given a different name.
Clear alarm with NC Start and continue processing.

12180 **Channel %1 block %2 illegal chaining of operators %3**
Explanation: %1 = Channel number
%2 = Block number, label
%3 = Chained operators
Operator chaining means the writing in sequence of binary and unary operators without using any form of parentheses.
Example:
N10 ERG = VARA - (- VARB) ; correct notation
N10 ERG = VARA - - VARB ; error !
Reaction: Alarm display. Interface signals are set. Correction block.
Formulate the expression correctly and unambiguously making use of parentheses. This improves clarity and readability of the program.
Clear alarm with NC Start and continue processing.

12190 **Channel %1 block %2 variable of type ARRAY has too many dimensions**
Explanation: %1 = Channel number
%2 = Block number, label
Array with variables of type STRING may be no more than 1-dimensional, and with all other variables no more than 2-dimensional.
Reaction: Alarm display. Interface signals are set. Correction block.
Remedy: Press the NC Stop key and select the function „Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. Correct the array definition, with multi-dimensional arrays define a second 2-dimensional array if necessary and operate it with the same field index.
Clear alarm with NC Start and continue processing.

12200 **Channel %1 block %2 symbol %3 cannot be created**
Explanation: %1 = Channel number
%2 = Block number, label
%3 = Symbol in the source block

emco

The symbol to be created with the DEF instruction cannot be created because:
• it has already been defined (e.g. as variable or function)
• the internal memory location is no longer sufficient (e.g. with large arrays).

Reaction: Alarm display. Interface signals are set. Correction block.
Remedy: Make the following checks:
• Check with the text editor whether the name to be allocated in the active program cycle (main program and called subprograms) has already been used.
• Estimate the memory requirements for the symbols already defined and reduce these if necessary by using fewer global and more local variables.
Clear alarm with NC Start and continue processing.

12210   **Channel %1 block %2 string %3 too long**
Explanation: %1 = Channel number
%2 = Block number, label
%3 = String in the source block
• In the definition of a variable of type STRING, it has been attempted to initialize more than 100 characters.
• In an allocation, it has been found that the string does not fit in the given variable.

Reaction: Alarm display. Interface signals are set. Correction block.
Remedy: Press the NC Stop key and select the function „Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block.
• Select shorter string or divide up the character string into 2 strings
• Define larger string variable
Clear alarm with NC Start and continue processing.

12220   **Channel %1 block %2 binary constant %3 in string too long**
Explanation: %1 = Channel number
%2 = Block number, label
%3 = Binary constant
When initializing or allocating the value of a variable of type STRING more than 8 bits have been found as binary constant. DEF STRING[8] OTTO = "ABC'H55''B000011111'DEF"

Reaction: Alarm display. Interface signals are set. Correction block.
Remedy: Press the NC Stop key and select the function „Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. In the window for the alarm message, the first characters of the binary constant are always displayed although the surplus bit might not be located until further on. Therefore, the **complete binary constant** must always be checked for an incorrect value.
Clear alarm with NC Start and continue processing.

12230   **Channel %1 block %2 hexadecimal constant %3 in string too long**
Explanation: %1 = Channel number
%2 = Block number, label
%3 = Hexadecimal constant
A string can also contain bytes that do not correspond to a character that can be entered or one that is available on a keyboard with a minimized number of keys. These characters can be input as binary or hexadecimal constants. They may occupy up to 1 byte each only - therefore be <256, e.g.
N10 DEF STRING[2] OTTO=" 'HCA' 'HFE' "

Reaction: Alarm display. Interface signals are set. Correction block.
Remedy: Press the NC Stop key and select the function „Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. In the window for the alarm message, the first characters of the binary constant are always displayed although the surplus bit might not be located until further on. Therefore, the complete **hexadecimal constant** must always be checked for an incorrect value.
Clear alarm with NC Start and continue processing.

12240   **Channel %1 block %2 tool orientation %3 defined repeatedly**
Explanation: %1 = Channel number
%2 = Block number, label
%3 = Text
Only one tool orientation can be programmed per DIN block. This can either be defined via the 3 Euler angles, or the end points of the axes, or through direction vectors.

Reaction: Alarm display. Interface signals are set. Correction block.
Remedy: Press the NC Stop key and select the function „Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. Since the tool orientation can be set in 3 different ways, the most advantageous should be selected. For specifying in this way, the addresses and value assignments must be programmed and all other orientation parameters must be removed.
Axis end points (additional axes): A, B, C axis identifiers,
Euler angles: A2, B2, C2
Direction vectors: A3, B3, C3
Clear alarm with NC Start and continue processing.

12250   **Channel %1 block %2 do not nest macro %3**
Explanation: %1 = Channel number
%2 = Block number, label
%3 = Source string

emco

The macro technique supplies a 1-line instruction or series of instructions with a new identifier by means of the keyword DEFINE. No further macro may be contained in the string of instructions (nesting).
Example:
N10 **DEFINE** MACRO1 **AS** G01 G91 X123 MACRO2 F100

Reaction: Alarm display. Interface signals are set. Correction block.
Remedy: Press the NC Stop key and select the function „Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. Nested macros must be replaced by the full program information. Clear alarm with NC Start and continue processing.

12260 **Channel %1 block %2 too many initialization values given for %3**
Explanation: %1 = Channel number
%2 = Block number, label
%3 = Source string
In the initialization of an array (array definition and value assignments to individual array elements) there are more initialization values than array elements.
Example:
N10 DEF INT OTTO[2,3]=(..., ..., {more than 6 values})

Reaction: Alarm display. Interface signals are set. Correction block.
Remedy: Press the NC Stop key and select the function „Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block.
Check the NC program to establish whether:
1. During array definition the number of array elements (n,m) was indicated correctly (DEF INT FIELDNAME[n,m] e.g. an array with 2 lines and 3 columns: n=2, m=3).
2. During initialization the value assignments have been made correctly (values of the individual field elements separated by **comma**, **decimal point** for variables of the type REAL)
Clear alarm with NC Start and continue processing.

12270 **Channel %1 block %2 macro identifier %3 already defined**
Explanation: %1 = Channel number
%2 = Block number, label
%3 = Source string macro name
The name of the macro to be selected by the instruction DEFINE is already defined in the control as:
Macro name
Keyword
Variable
Configured identifier.

Reaction: Alarm display. Interface signals are set. Correction block.
Remedy: Press the NC Stop key and select the function „Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block.
Select DEFINE instruction with another macro name.
Clear alarm with NC Start and continue processing.

12290 **Channel %1 block %2 arithmetic variable % 3 not defined**
Explanation: %1 = Channel number
%2 = Block number, label
%3 = Source string arithmetic variable
Only the R parameters are predefined as arithmetic variables. All other arithmetic variables must be defined with the DEF instruction before being used. The number of arithmetic parameters is defined via machine data. The names must be unambiguous and may not be repeated in the control (exception: local variables).

Reaction: Alarm display. Interface signals are set. Correction block.
Remedy: Press the NC Stop key and select the function „Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. Define the required variable in the definition part of the program (possibly in the calling program if it is to be a global variable).
Clear alarm with NC Start and continue processing.

12300 **Channel %1 block %2 call-by-reference argument missing on subroutine call %3**
Explanation: %1 = Channel number
%2 = Block number, label
%3 = Source string
In the subroutine definition, a formal **REF parameter (call-by-reference parameter)** has been specified with no actual parameter assigned to it. The assignment takes place in the subroutine call on the basis of the position of the variable name and not on the basis of the name.
Example:
Subroutine: (2 call-by-value parameters X and Y, 1 call-by-reference parameter Z)
PROC XYZ (INT X, INT Y, VAR INT Z)
:
M17
ENDPROC
Main program:
N10 DEF INT X
N11 DEF INT Y
N11 DEF INT Z

:
N50 XYZ (X, Y) ; REF parameter Z missing!
or
N50 XYZ (X, Z) ; REF parameter Z missing!

Reaction: Alarm display. Interface signals are set. Correction block.

Remedy: Press the NC Stop key and select the function „Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. Assign a variable to all REF parameters (call-by-reference parameters) of the subroutine when calling. No variable must be assigned to „normal" formal parameters (call-by-value parameters), as these are defaulted with 0.
Clear alarm with NC Start and continue processing.

12310   **Channel %1 block %2 axis argument missing on procedure call %3**

Explanation: %1 = Channel number
%2 = Block number, label
%3 = Source string
When calling the subroutine, an AXIS parameter is missing which, according to the EXTERN declaration, should be present. With the EXTERN instruction, user-defined subroutines (procedures) are made „known" that have a parameter transfer. Procedures without parameter transfer require no EXTERN declaration.
Example:
Subroutine XYZ (with the formal parameters):
PROC XYZ (INT X, VAR INT Y, AXIS A, AXIS B)
EXTERN instruction (with variable types):
EXTERN XYZ (INT, VAR INT, AXIS, AXIS)
Subroutine call (with actual parameters):
N10 XYZ (, Y1, R_TABLE)
Variable X is defaulted with value 0
Variable Y is supplied with the value of the variable Y1 and returns the results to the calling program after the subroutine run
Variable A is supplied with the axis in R_TABLE
Variable B missing!

Reaction: Alarm display. Interface signals are set. Correction block.

Remedy: Press the NC Stop key and select the function „Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. Program the missing AXIS parameter in the call.
Clear alarm with NC Start and continue processing.

12320   **Channel %1 block %2 argument %3 must be call-by-reference**

Explanation: %1 = Channel number
%2 = Block number, label
%3 = Source string
A constant or the result of a mathematical expression has been assigned to a REF parameter instead of a variable at the time of the subroutine call, even though only variable identifiers are allowed.
Examples:
N10 XYZ (NAME_1, **10,** OTTO) or N10 XYZ (NAME_1, **5 + ANNA,** OTTO)

Reaction: Alarm display. Interface signals are set. Correction block.

Remedy: Press the NC Stop key and select the function „Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block.
Remove the constant or the mathematical expression from the NC block.
Clear alarm with NC Start and continue processing.

12330   **Channel %1 block %2 type of argument %3 incorrect**

Explanation: %1 = Channel number
%2 = Block number, label
%3 = Source string
When calling a procedure (a subroutine) it is found that the type of the actual parameter cannot be converted into the type of the formal parameter. There are two possible cases:
• **Call-by-reference parameter:** Actual parameter and formal parameter must be of precisely the same type, e.g. STRING, STRING.
• **Call-by-value parameter:** Actual parameter and formal parameter can in principle be different providing conversion is basically possible. In the present case, however, the types are generally not compatible, e.g. STRING-REAL.

from - to	REAL	INT	BOOL	CHAR	STRING	AXIS	FRAME
REAL	yes	yes*	yes**	yes*	-	-	-
INT	yes	yes	yes**	if value 0..255	-	-	-
BOOL	yes	yes	yes	yes	-	-	-
CHAR	yes	yes	yes**	yes	yes	-	-
STRING	-	-	yes***	only if 1 character	yes	-	-
AXIS	-	-	-	-	-	yes	-
FRAME	-	-	-	-	-	-	yes

*   At type conversation from REAL to INT fractional values that are >=0.5 are rounded up, others are rounded down

	** Value <>0 corresponds to TRUE, value ==0 corresponds to FALSE.
	*** String length 0 => FALSE, otherwise TRUE
Reaction:	Alarm display. Interface signals are set. Correction block.
Remedy:	Press the NC Stop key and select the function „Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. Check transfer parameters of the subroutine call and define the application accordingly as call-by-value or call-by-reference parameter. Clear alarm with NC Start and continue processing.

12340   **Channel %1 block %2 number of arguments exceeded in %3**

Explanation:
%1 = Channel number
%2 = Block number, label
%3 = Source string
When calling a function or a procedure (predefined or user-defined) more parameters were transferred than defined.
Predefined functions and procedures:
The number of parameters has been set permanently in the NCK.
User-defined functions and procedures:
The number of parameters is established by type and name in the definition

Reaction: Alarm display. Interface signals are set. Correction block.

Remedy: Press the NC Stop key and select the function „Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. Check whether the correct procedure/function has been called. Program the number of parameters in accordance with the procedure/function.
Clear alarm with NC Start and continue processing.

12350   **Channel %1 block %2 argument %3 not accepted because AXIS argument is missing**

Explanation:
%1 = Channel number
%2 = Block number, label
%3 = Source string
An attempt has been made to transfer actual parameters although axis parameters located before them have not been assigned. For procedure or function calls, assignment of parameters that are no longer required can be omitted, if **subsequently no further** parameters are to be transferred.
Example:
N10 FGROUP(X, Y, Z, A, B) ; max. 8 axes possible
The following call-by-value parameters would then be defaulted with zero because the space-dependent assignment has been lost on account of the omitted axis parameters. Axes that can be omitted and following parameters do not occur in the predefined procedures and functions.

Reaction: Alarm display. Interface signals are set. Correction block.

Remedy: Press the NC Stop key and select the function „Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on theincorrect block. In **predefined procedures and functions** either remove the following parameters or transfer any preceding axis parameters. In **user-defined procedures and functions,** parameter transfer must be programmed in accordance with the instructions given in the machine manufacturer's programming guide.
Clear alarm with NC Start and continue processing.

12360   **Channel %1 block %2 dimension of argument %3 incorrect**

Explanation:
%1 = Channel number
%2 = Block number, label
%3 = Source string
The following possibilities of error must be checked:
1. The current parameter is an array, but the formal parameter is a variable
2. The current parameter is a variable, but the formal parameter is an array
3. The current and formal parameters are arrays, but not with the dimensions to be defined.

Reaction: Alarm display. Interface signals are set. Correction block.

Remedy: Press the NC Stop key and select the function „Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block.
Correct the NC part program in accordance with the cause of error as listed above.
Clear alarm with NC Start and continue processing.

12370   **Channel %1 block %2 range of values exceeded for %3**

Explanation:
%1 = Channel number
%2 = Block number, label
%3 = Source string
Outside of an initialization block, a variable has been provided with a value range. The definition of program-global variables is allowed only in special initialization blocks. They can be provided with a value range.

Reaction: Alarm display. Interface signals are set. Correction block.

Remedy: Press the NC Stop key and select the function „Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. Remove specification of value range (begins with the keyword OF) or define the variable as global variable in the initialization block and provide it with a value range.
Clear alarm with NC Start and continue processing.

12390   **Channel %1 block %2 type of initial value for %3 cannot be converted**
Explanation: %1 = Channel number
%2 = Block number, label
%3 = Source string
During initialization, a value has been assigned to a variable that does not correspond to the type of the variable, nor can it be converted to the data type of the variable.

from - to	REAL	INT	BOOL	CHAR	STRING
REAL		yes*	yes	yes**	-
INT	yes		yes	yes**	-
BOOL	yes	yes		yes	-
CHAR	yes	yes	yes**		yes
STRING	-	-	yes	yes***	

*   Value <>0 corresponds to TRUE, value ==0 corresponds to FALSE.
**   String length 0 => FALSE, otherwise TRUE
***   If only one character
It is not possible to convert from type AXIS and FRAME nor into type AXIS and FRAME.
Reaction: Alarm display. Interface signals are set. Correction block.
Remedy: Press the NC Stop key and select the function „Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block.
• Define variable type such that the initialization value can be assigned, or
• Select initialization value in accordance with the variable definition.
Clear alarm with NC Start and continue processing.

12400   **Channel %1 block %2 element of array %3 does not exist**
Explanation: %1 = Channel number
%2 = Block number, label
%3 = Source string
The following causes are possible:
- Impermissible index list; an axis index is missing
- Array index does not match the definition of the variables
- An attempt was made to access a variable at array initialization via SET or REP; this attempt did not correspond to the standard access. Single character access, partial frame access, omitted indices not possible.
A nonexistent element was addressed on initializing this array.
Reaction: Alarm display. Interface signals are set. Correction block.
Remedy: Press the NC Stop key and select the function „Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block.
• **Array initialization:** Check the array index of the addressed element. The 1st array element is given the index [0,0], the 2nd array element [0,1] etc. The right array index (column index) is incremented first. In the 2nd row, the 4th element is also addressed with the index [1,3] (the indices start at zero).
• **Array definition:** Check the size of the array. The 1st number indicates the number of elements in the 1st dimension (number of rows), the 2nd number indicates the number of elements in the 2nd dimension (number of columns). An array with 2 rows and 3 columns must be defined by specifying [2,3]..
Clear alarm with NC Start and continue processing.

12410   **Channel %1 block %2 incorrect index type for %3**
Explanation: %1 = Channel number
%2 = Block number, label
%3 = Source string
In assigning a value to an element of an array variable, the array index was specified in a way that is not allowed. Only the following are allowed as array index (in square brackets):
• **Axis identifier,** provided the array variable was defined as data type FRAME.
• **Integer values** for all other data types.
Reaction: Alarm display. Interface signals are set. Correction block.
Remedy: Press the NC Stop key and select the function „Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block.
Correct indices of the array element with respect to variable definition or define the array variable differently.
Clear alarm with NC Start and continue processing.

12420   **Channel %1 block %2 identifier %3 too long**
Explanation: %1 = Channel number
%2 = Block number, label
The symbol to be defined or the specified jump target has a name which is longer than the 32 characters allowed.
Reaction: Alarm display. Interface signals are set. Correction block.
Remedy: Press the NC Stop key and select the function „Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. The symbol to be created or the target of program jumps (label) must be selected within the system agreements, that means the name must begin with 2 letters (but the 1st sign must not be "§") and may be up to a maximum of 32 characters.
Clear alarm with NC Start and continue processing.

emco

12430   **Channel %1 block %2 invalid index**
Explanation: %1 = Channel number
%2 = Block number, label
In specifying an array index (in the array definition) an index was used that is outside the permissible range.
Reaction: Alarm display. Interface signals are set. Correction block.
Remedy: Press the NC Stop key and select the function „Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block.
Specify array index within the permissible range. Value range per array dimension: 1 - 32 767.
Clear alarm with NC Start and continue processing.

12440   **Channel %1 block %2 maximum number of formal arguments exceeded**
Explanation: %1 = Channel number
%2 = Block number, label
In the definition of a procedure (a subroutine) or in an EXTERN instruction, more than 127 formal parameters have been specified.
Example:
PROC ABC (FORMPARA1, FORMPARA2, ... FORMPARA127, **FORMPARA128, ...**)
EXTERN ABC (FORMPARA1, FORMPARA2, ... FORMPARA127, **FORMPARA128, ...**)
Reaction: Alarm display. Interface signals are set. Correction block.
Remedy: Press the NC Stop key and select the function „Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. A check must be made to determine whether all parameters really have to be transferred. If so, the formal parameters can be reduced by using global variables or R parameters, or by grouping together parameters of the same type to form an array and transfer them in this form.
Clear alarm with NC Start and continue processing.

12450   **Channel %1 block %2 label defined repeatedly**
Explanation: %1 = Channel number
%2 = Block number, label
The label of this block already exists.
If the NC program is compiled off-line, the entire program is compiled block for block. During this procedure all multiple labels are recognized; this is not always the case with **on-line compilation**. (Only the actual program run is compiled here, i.e. program branches that are not passed through in this run are disregarded and could therefore contain programming errors.)
Reaction: Alarm display. Interface signals are set. Correction block.
Remedy: Press the NC Stop key and select the function „Correction block" with the softkey PROGRAM CORRECT. The correction pointer is positioned on the block where the displayed label occurs for the second time. Use the editor to search the part program where this label occurs for the first time, and change one of the names.
Clear alarm with NC Start and continue processing.

12460   **Channel %1 block %2 maximum number of symbols exceeded with %3**
Explanation: %1 = Channel number
%2 = Block number, label
%3 = Source string
The max. number of variable definitions (GUD, LUD), macro definitions, cycle programs, cycle parameters, that the controller's data management is able to handle, has been exceeded. If this alarm occurs in conjunction with alarm 15180 (initial.ini download failed), then this alarm shows the name of the block causing the error. (For a list of names and their meaning, please refer to alarm 6010)
Reaction: Alarm display. Interface signals are set. Correction block.
Remedy: Reduce the symbols in the block (possibly by using the array technique or by using R parameters), or adapt the machine data (if you have access rights). $MC_MM_NUM_LUD_NAMES_TOTAL with error in LUD blocks (i.e. if more variable definitions were made in the active part programs than allowed by the MD). GUD data blocks can only cause errors as part of the 'initial.ini download' process. Macros and cycle program definitions are reloaded at each POWER ON/ NCK-RESET. This means that these blocks can only cause errors in conjunction with this process.
See also the explanations for alarm 6010
Clear alarm with NC Start and continue processing.

12470   **Channel %1 block %2 unknown G function %3 used**
Explanation: %1 = Channel number
%2 = Block number, label
%3 = Source string
In the displayed block, a non-defined G function has been programmed. Only "real" G functions are checked, which begin with the address G, e.g. G555. "Named" G functions such as CSPLINE, BRISK etc. are interpreted as subroutine names.
Reaction: Alarm display. Interface signals are set. Correction block.
Remedy: Press the NC Stop key and select the function „Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. It must be decided on the basis of the machine manufacturer's programming guide as to whether or not the displayed G function is always omitted or not possible, or whether a standard G function has been reconfigured (or introduced by OEM). Remove G function from the part program or program function call in accordance with the machine manufacturer's programming guide.

emco

Clear alarm with NC Start and continue processing.

12480　　　　　　**Channel %1 block %2 subroutine %3 already defined**
Explanation:　　　　%1 = Channel number
　　　　　　　　　　%2 = Block number, label
　　　　　　　　　　%3 = Source string
　　　　　　　　　　The name used in the PROC or EXTERN instruction has already been defined in another call description (e.g. for cycles).
　　　　　　　　　　Example:
　　　　　　　　　　EXTERN **CYCLE85** (VAR TYP1, VAR TYP2, ...)
Reaction:　　　　　Alarm display. Interface signals are set. Correction block.
Remedy:　　　　　Press the NC Stop key and select the function „Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. A program name must be selected that has not yet been used as identifier (theoretically, the parameter declaration of the EXTERN instruction could also be adapted to the existing subroutine in order to avoid the alarm output. However, it would have been identically defined twice).
　　　　　　　　　　Clear alarm with NC Start and continue processing.

12520　　　　　　**Channel %1 too many machine data %3 in block %2**
Explanation:　　　　%1 = Channel number
　　　　　　　　　　%2 = Block number, label
　　　　　　　　　　%3 = Source symbol
　　　　　　　　　　In the part program, in the machine data file (..._TOA) and in the initialization file (..._INI), no more than 2 machine data may be used per block.
　　　　　　　　　　Example:
　　　　　　　　　　N ...
　　　　　　　　　　N 100　$TC_DP1 [5,1] = 130, $TC_DP3 [5,1] = 150.123, $TC_DP4 [5,1] = 223.4,
　　　　　　　　　　　　　　$TC_DP5 [5,1] = 200.12, $TC_DP6 [5,1] = 55.02
　　　　　　　　　　N ...
Reaction:　　　　　Alarm display. Interface signals are set. Correction block.
Remedy:　　　　　Press the NC Stop key and select the function „Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block.
　　　　　　　　　　• Divide up the part program block into several blocks
　　　　　　　　　　•　If necessary, use the local variable for storing intermediate results
　　　　　　　　　　Clear alarm with NC Start and continue processing.

12530　　　　　　**Channel %1 block %2 invalid index for %3**
Explanation:　　　　%1 = Channel number
　　　　　　　　　　%2 = Block number, label
　　　　　　　　　　%3 = Source string
　　　　　　　　　　In macro definitions, an attempt was made to define a G function with more than 3 decades or an M function with more than 2 decades as identifier of the macro.
　　　　　　　　　　Example:
　　　　　　　　　　_N_UMAC_DEF DEFINE **G4444** AS G01 G91 G1234
　　　　　　　　　　DEFINE **M333** AS M03 M50 M99
　　　　　　　　　　:
　　　　　　　　　　M17
Reaction:　　　　　Alarm display. Interface signals are set. Correction block.
Remedy:　　　　　Press the NC Stop key and select the function „Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. Modify the macro definition in accordance with the Programming Guide.
　　　　　　　　　　Clear alarm with NC Start and continue processing.

12540　　　　　　**Channel %1 block %2 is too long or too complex**
Explanation:　　　　%1 = Channel number
　　　　　　　　　　%2 = Block number, label
　　　　　　　　　　The maximum internal block length after translator processing must not exceed 256 characters. After editing, for example, several macros in the block or a multiple nesting, this limit can be exceeded.
Reaction:　　　　　Alarm display. Interface signals are set. Correction block.
Remedy:　　　　　Press the NC Stop key and select the function „Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block.
　　　　　　　　　　Divide up the program block into several subblocks.
　　　　　　　　　　Clear alarm with NC Start and continue processing.

12550　　　　　　**Channel %1 block %2 identifier %3 not defined or option does not exist**
Explanation:　　　　%1 = Channel number
　　　　　　　　　　%2 = Block number, label
　　　　　　　　　　%3 = Source symbol
　　　　　　　　　　The displayed identifier was not defined before being used.
　　　　　　　　　　Macro: Keyword, to be defined with the DEFINE ... AS ... instruction is missing in one of the files:
　　　　　　　　　　_N_SMAC_DEF, _N_MMAC_DEF, _N_UMAC_DEF, _N_SGUD_DEF, _N_MGUD_DEF, _N_UGUD_DEF
　　　　　　　　　　Variable: DEF instruction missing
　　　　　　　　　　Program: PROC declaration missing

emco

Reaction:	Alarm display. Interface signals are set. Correction block.
Remedy:	Press the NC Stop key and select the function „Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. - Correct the names used (typing error) - Check the definition of variables, subroutines and macros - Check options. Clear alarm with NC Start and continue processing.

12560   **Channel %1 block %2 programmed value %3 exceeds allowed limits**

Explanation:	%1 = Channel number %2 = Block number, label %3 = Source string In a value assignment, the permissible value range of the data type has been exceeded.
Reaction:	Alarm display. Interface signals are set. Correction block.
Remedy:	Press the NC Stop key and select the function „Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. Assign value within the value range of the various data types, or if necessary use another type in order to increase the size of the value range, e.g. INT -> REAL.

Variable type	Property	Value range
REAL	Fractional number with dec. pt.	$\pm(2^{-1022} - 2^{+1023})$
INT	Integers with signs	$\pm (2^{31} -1)$O
BOOL	Truth value TRUE, FALSE	0,1
CHAR	1 ASCII character	0 - 255
STRING	Character string (max. 100 values)	0 - 255
AXIS	Axis addresses	**Axis names only**
FRAME	Geometric information	**As for axis paths**

Clear alarm with NC Start and continue processing.

12600   **Channel %1 block %2 invalid checksum of line**

Explanation:	%1 = Channel number %2 = Block number On processing an INI file or when executing a TEA file, an invalid line checksum has been detected.
Reaction:	Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy:	Correct INI file or correct MD and create new INI file (via „upload"). Steuerung AUS - EIN schalten.

12610   **Channel %1 block %2 accessing single char with call-by-reference argument not allowed %3**

Explanation:	%1 = Channel number %2 = Block number, label %3 = Source string An attempt has been made to use a single character access for a call-by-reference parameter.
Reaction:	Alarm display. Interface signals are set. Correction block.
Remedy:	Temporarily store single characters in user-defined CHAR variable and transfer this. Clear alarm with NC Start and continue processing.

12620   **Channel %1 block %2 accessing this variable as single char not allowed %3**

Explanation:	%1 = Channel number %2 = Block number, label %3 = Source string The variable is not a user-defined variable. The single character access is only allowed for user-defined variables (LUD/GUD).
Reaction:	Alarm display. Interface signals are set. Correction block.
Remedy:	Temporarily store variable in user-defined STRING, process this and put back into storage. Clear alarm with NC Start and continue processing.

12630   **Channel %1 block %2 skip / label not allowed**

Explanation:	%1 = Channel number %2 = Block number Blocks with control structures (FOR, ENDIF, etc.) cannot be concealed and must not contain any labels.
Reaction:	Alarm display. Interface signals are set. Correction block.
Remedy:	Teileprogramm korrigieren: Correct part program: Create conceal identifier by IF testing and write label on in its own in the block in front of the control structure block. Clear alarm with NC Start and continue processing.

12640   **Channel %1 block %2 invalid nesting of control structures**

Explanation:	%1 = Channel number %2 = Block number Error in program run: Opened control structures (IF-ELSE-ENDIF, LOOP-ENDLOOP etc.) are not terminated or there is no beginning of loop for the programmed end of loop. Example: LOOP ENDIF ENDLOOP
Reaction:	Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy:	Correct part program in such a way that all opened control structures are also terminated. Clear alarm with RESET key. Restart part program.

emco

12641   **Channel %1 block %2 nesting level of control structures exceeds limit**
Explanation: %1 = Channel number
                %2 = Block number
                Max. nesting depth control structures (IF-ELSE-ENDIF, LOOP-ENDLOOP etc.) exceeded. At the present time, the
                max. nesting depth is 8
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Correct part program. If necessary, move parts to a subroutine.
                Clear alarm with RESET key. Restart part program.

12650   **Channel %1 block %2 axis %3 name different in channel %4**
Explanation: %1 = Channel number
                %2 = Block number
                %3 = Source symbol
                %4 = Channel number with different axis definition
                In cycles that are preprocessed at Power On, only those geometry and channel axis identifiers may be used that
                exist in all channels with the same meaning. In different channels, different axis indices are assigned to the axis
                identifier. The axis identifiers are defined via machine data 20060 AXCONF_GEOAX_NAME_TAB and 20080
                AXCONF_CHANAX_NAME_TAB.
                Example: C is the 4th channel axis in channel 1 and the 5th channel axis in channel 2.
                If the axis identifier C is used in a cycle that is preprocessed at Power On, then this alarm is issued.
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: 1. Modify machine data: Select the same identifiers for geometry and channel axes in all channels. Example: The
                geometry axes are called X, Y, Z in all channels. They can then also be programmed directly in preprocessed
                channels.
                PROC DRILL G1 Z10 F1000 M17 or
                2. Do not program the axis directly in the cycle but define it as parameter of the Axis type. Example: Cycle definition:
                PROC DRILL (AXIS DRILLAXIS) G1 AX[DRILLAXIS]=10 F1000 M17
                Call from the main program:
                DRILL(Z)
                Clear alarm with RESET key. Restart part program.

12661   **Channel %1 block %2 technology cycle %3: no further program call possible**
Explanation: %1 = Channel number
                %2 = Block number
                %3 = Name of the technology cycle call
                In a technology cycle it is not possible to call a subroutine or another technology cycle.
Reaction: Alarm display. Interface signals are set. Correction block
Remedy: Modify part program.
                Clear alarm with the RESET key.

14000   **Channel %1 block %2 Unzulaessiges Dateiende**
Explanation: %1 = Channel number
                %2 = Block number, label
                Als Dateiende von Hauptprogrammen wird ein **M02** oder ein **M30** erwartet, von Unterprogrammen **M17.** Von der
                Satzaufbereitung (Datenhaltung) wird kein Folgesatz geliefert, obwohl im vorhergehenden block kein Dateiende
                programmiert war.
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Kontrollieren, ob das Programmende vergessen wurde einzugeben, oder ob im letzten Programmsatz ein Sprung auf
                einen Programmabschnitt, in dem die Endekennung steht, erfolgt.
                Clear alarm with RESET key. Restart part program.

14001   **Channel %1 block %2 error at end of file, line feed missing**
Explanation: %1 = Channel number
                %2 = Block number, label
                After system-internal data manipulation (e.g. when transferring blocks from an external source) a subfile can end
                without having LF as the last character.
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Read out the part program, modify it with a text editor (e.g., insert blanks or comments before the displayed block),
                so that after reading it in again the part program has a different structure in the memory.
                Clear alarm with RESET key. Restart part program.

14010   **Channel %1 block %2 invalid default argument in subroutine call**
Explanation: %1 = Channel number
                %2 = Block number, label
                In a subroutine call with parameter transfer, parameters have been omitted that cannot be replaced by default
                parameters (call-by-reference parameters or parameters of type AXIS. The other missing parameters are defaulted
                with the value 0 or with the unit frame in the case of frames).
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: The missing parameters must be provided with values in the subroutine call.
                Clear alarm with RESET key. Restart part program.

emco

14011   **Channel %1 block %2 program %3 not existing or not released for machining**

Explanation: %1 = Channel number
%2 = Block number, label
%3 = Program name
An unknown identifier (string) was found in the part program. It is therefore assumed that this is a program name. The part program indicated in a subprogram call or SETINT statement does not exist or is not released for machining.

Reaction: Alarm display. Interface signals are set. Correction block is reorganized.

Remedy: The alarm may have different causes:
- Typing error of the identifier stated in parameter 3
- Check subprogram call / SETINT statement or PROC statement. Reload part program and release for machining.
- Parameter 3 can be a macro name. The macro definition file has an inappropriate content or it is not stored in the directory DEF_DIR or it has not been set active (via POWERON or via MMC operating step or by PI service 'F_COPY').
- Parameter 3 can be a GUD variable. There is no GUD definition file defining the variable or it is not stored in the directory DEF_DIR or it has not been set active (via the INITIAL_INI procedure or via MMC operating step or by PI service 'F_COPY').
Clear alarm with NC Start and continue program.

14012   **Channel %1 block %2 lowest subroutine level exceeded**

Explanation: %1 = Channel number
%2 = Block number, label
The maximum nesting depth of 8 program levels has been exceeded. Subroutines can be called from the main program, and these in turn may have a nesting depth of 7..
In **interrupt routines** the maximum number of levels is 4!

Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.

Remedy: Modify the machining program so that the nesting depth is reduced, e.g. using the editor copy a subroutine of the next nesting level into the calling program and remove the call for this subroutine. This reduces the nesting depth by one program level.
Clear alarm with RESET key. Restart part program.

14013   **Channel %1 block %2 number of subroutine passes invalid**

Explanation: %1 = Channel number
%2 = Block number, label
In a subroutine call the programmed number of passes P is zero or negative.

Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.

Remedy: Program number of passes between 1 and 9 999.
Clear alarm with RESET key. Restart part program.

14014   **Channel %1 block %2 selected program %3 or access permission not available**

Explanation: %1 = Channel number
The selected part program is not in the NCK memory or it is the access authorization for program selection at a higher level corresponding to the present status of the control. When this program was generated, it received the protection level that was active at the time for the NC control.

Reaction: Alarm display.

Remedy: Transfer the required program into the NCK memory or check the name of the directory (workpiece overview) and of the program (program overview) and correct these.
Increase the present protection level to at least the level of the program being executed (by password input).
Clear alarm with the Cancel key. No further operator action necessary.

14015   **Channel %1: no access permission for file**

Explanation: %1 = Channel number
A program is to be executed for which the current protection level is too low. When this program was generated, it received the protection level that was active at the time for the NC control.

Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.

Remedy: Increase the present protection level to at least the level of the program being executed (by password input).
Clear alarm with RESET key. Restart part program.

14020   **Channel %1 block %2 wrong number of arguments on function or procedure call**

Explanation: %1 = Channel number
%2 = Block number, label
When a predefined function or procedure (subroutine) was called, the number of actual parameters was either
• programmed **basically incorrectly**, e.g. in frames an odd number of parameters (except when mirroring), or
• **too few parameters** were transferred. (Too many parameters are already recognized in the compiler, which then triggers alarm 11 039: "Channel %1 block %2 parameter number too large").

Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.

Remedy: Correct the number of transfer parameters in the call in the NC block.
Clear alarm with RESET key. Restart part program.

14021   **Channel %1 block %2 wrong number of arguments on function or procedure call**

Explanation: %1 = Channel number
%2 = Block number, label
In a function or procedure call, an impermissible number of actual parameters has been programmed.

Reaction:	Alarm display. Interface signals are set. Interpreter stop. NC Start disable. Modify part program. Clear alarm with RESET key. Restart part program.

14040 **Channel %1 block %2 error in end point of circle**
Explanation:
%1 = Channel number
%2 = Block number, label
In circular interpolation, either the circle radii for the initial point and the end point are further apart, or the circle center points are further apart, than specified in the machine data.
1. In circle radius programming the starting and end points are identical, thus the circle position is not determined by starting and end points.
2. **Radii:** The NCK calculates from the present start point and the other programmed circle parameters the radii for the start and the end point. An alarm message is issued if the difference between the circle radii is either greater than the value in the MD 21000 **CIRCLE_ERROR_CONST** (for small radii, if the programmed radius is smaller than the quotient of the machine data **CIRCLE_ERROR_CONST** divided by 21010 **CIRCLE_ERROR_FACTOR),** or greater than the programmed radius multiplied by the MD **CIRCLE_ERROR_FACTOR** (for large radii, if the programmed radius is greater than the quotient of the machine data **CIRCLE_ERROR_CONST** divided by **CIRCLE_ERROR_FACTOR).**
3. **Center points:** A new circle center is calculated using the circle radius at the starting position. It lies on the mid-perpendicular positioned on the connecting straight line from the starting point to the end point of the circle. The angle in the radian measure between both straight lines from the starting point to the center calculated/programmed as such must be lower than the root of 0.001 (corresponding to approx. 1.8 degrees).

Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Check MD 21000 **CIRCLE_ERROR_CONST** and 21010 **CIRCLE_ERROR_FACTOR**. If the values are within reasonable limits, the circle end point or the circle mid-point of the part program block must be programmed with greater accuracy.
Clear alarm with RESET key. Restart part program.

14045 **Channel %1 block %2 error in tangent circle programming**
Explanation:
%1 = Channel number
%2 = Block number, label
The alarm may have the following causes:
- The tangent direction is not defined for tangent circle / e.g. because no other travel block has been programmed before the current block.
- No circle can be formed from start and end point as well as tangent direction because - seen from the start point
- the end point is located in the opposite direction to that indicated by the tangent.
- It is not possible to form a tangent circle since the tangent is located vertically to the active plane.
- In the special case in which the tangent circle changes to a straight line, several complete circular revolutions were programmed with TURN.

Reaction: Alarm display. Interface signals are set. Correction block is reorganized. NC Start disable. NC Stop when alarm at block end.
Remedy: Modify part program.
Clear alarm with NC Start and continue program.

14050 **Channel %1 block %2 nesting depth for arithmetic operations exceeded**
Explanation:
%1 = Channel number
%2 = Block number, label
For calculating arithmetic expressions in NC blocks, an operand stack with a fixed set size is used. In very complex expressions, this stack can overflow.

Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Divide up complex arithmetic expressions into several simpler arithmetic blocks.
Clear alarm with RESET key. Restart part program.

14051 **Channel %1 block %2 arithmetic error in part program**
Explanation:
%1 = Channel number
%2 = Block number, label
• In calculating an arithmetic expression, an overflow has occurred (e.g. division by zero).
• In a data type, the representable value range has been exceeded

Reaction: Alarm display. Interface signals are set. Correction block is reorganized.
Remedy: Analyze the program and correct the defective point in the program.
Clear alarm with NC Start and continue program.

14060 **Channel %1 block %2 invalid skip level with differential block skip**
Explanation:
%1 = Channel number
%2 = Block number, label
With „Differential block skip", a skip level greater than 7 has been specified (in packet 1 specification of a value for the skip level is rejected by the converter as a **syntax error**, i.e. the only possibility is a "Suppress block" ON/OFF on one level).

Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Enter a skip level (number behind the slash) less than 8.
Clear alarm with RESET key. Restart part program.

14070 **Channel %1 block %2 memory for variables not sufficient for subroutine call**
Explanation:
%1 = Channel number
%2 = Block number, label
A called subroutine cannot be processed (opened), either because the internal data memory to be created for

emco

general purposes is not large enough, or because the available memory for the local program variables is too small. The alarm can only occur in MDA mode.

Reaction:	Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy:	Abschnitt des Teileprogramms analysieren: 1. Has the most useful data type always been selected in the variable definitions? (For example REAL for data bits is poor; BOOL would be better) 2. Can local variables be replaced by global variables? Clear alarm with RESET key. Restart part program.

14080   **Channel %1 block %2 jump destination not found**

Explanation:	%1 = Channel number %2 = Block number, label In conditional and unconditional jumps, the jump destination within the program must be a block with a **label** (symbolic name instead of block number). If no jump destination has been found with the given label when searching **in the programmed direction**, an alarm is output.
Reaction:	Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy:	Check NC part program for the following possible errors: 1. Check whether the target designation is identical with the label. 2. Is the jump direction correct? 3. Has the label been terminated with a colon? Clear alarm with RESET key. Restart part program.

14090   **Channel %1 block %2 invalid D number**

Explanation:	%1 = Channel number %2 = Block number, label A value less than zero has been programmed under address D. A set of parameters with 25 correction values has been automatically assigned to each active tool. Each tool can have 9 sets of parameters (D1 - D9, initial setting is D1). When the D number changes, the new parameter set is active (D0 is used for deselecting the correction values). N10 G.. X... Y... T15 Parameter set D1 of T15 active N50 G.. X... D3 M.. Parameter set D3 of T15 active N60 G.. X.. T20 Parameter set D1 of T20 active
Reaction:	Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy:	Program D numbers in the permissible value range (D0, D1 to D9). Clear alarm with RESET key. Restart part program.

14091   **Channel %1 block %2 invalid function, index %3**

Explanation:	%1 = Channel number %2 = Block number, label Programming RET in the 1st program level.
Reaction:	Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy:	Select G functions in keeping with the possibilities provided by the NCK. Clear alarm with RESET key. Restart part program.

14092   **Channel %1 block %2 axis %3 has wrong axis type**

Explanation:	%1 = Channel number %2 = Block number, label %3 = Axis name, spindle number One of the following three programming errors has occurred: 1. The keyword **WAITP(x)** "Wait with block change until the specified positioning axis has reached its end point" has been used for an axis that is not a positioning axis. 2. **G74** "Reference point approach from the program" has been programmed for a spindle. (Only axis addresses are permitted.) 3. The keyword **POS/POSA** has been used for a spindle. (The keywords SPOS and SPOSA must be programmed for the spindle positions.)
Reaction:	Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy:	Correct the part program depending on which of the above errors is involved. Clear alarm with RESET key. Restart part program.

14093   **Channel %1 block %2 path interval zero or negative with polynominal interpolation**

Explanation:	%1 = Channel number %2 = Block number, label In the polynomial interpolation POLY, a negative value or zero has been programmed under the keyword for the polynomial length **PL=.**...
Reaction:	Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy:	Press the NC Stop key and select the function „Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block. Correct the value given in **PL = ...** Clear alarm with RESET key. Restart part program.

14094   **Channel %1 block %2 polynominal degree greater than 3 programmed for polynominal interpolation**

Explanation:	%1 = Channel number %2 = Block number, label The polynomial degree in the polynomial interpolation is based on the number of programmed coefficients for an axis. The maximum possible polynomial degree is 3, i.e. the axes are according to the function: $f(p) = a_0 + a_1 p + a_2 p_2 + a_3 p_3$ The coefficient a 0 is the actual position at the start of interpolation and is not programmed!
Reaction:	Alarm display. Interface signals are set. Interpreter stop. NC Start disable.

emco

Remedy:	Reduce the number of coefficients. The polynomial block may have a form no greater than the following: N1 POLY PO[X]=(1.11, 2.22, 3.33) PO[Y]=(1.11, 2.22, 3.33) N1 PO[n]=... PL=44 n ... axis identifier, max. 8 path axes per block Clear alarm with RESET key. Restart part program.

14095   **Channel %1 block %2 circle programmed with zero radius**

Explanation:	%1 = Channel number %2 = Block number, label The radius entered for radius programming is too small, i.e. the programmed radius is smaller than half of the distance between start and end point.
Reaction:	Alarm display. Interface signals are set. Correction block is reorganized.
Remedy:	Modify part program Clear alarm with NC Start and continue program.

14096   **Channel %1 block %2 type conversion not possible**

Explanation:	%1 = Channel number %2 = Block number, label During the program run, a variable value assignment or an arithmetic operation has caused data to be processed in such a way that they have to be converted to another type. This would lead to the value range being exceeded.

Variable type	Property	Value range
REAL	Fractional numbers with dec. pt.	$\pm(2^{-1022} - 2^{+1023})$
INT	Integers with signs	$\pm(2^{31} - 1)O$
BOOL	Truth value TRUE, FALSE	**0,1**
CHAR	1 ASCII character	**0 - 255**
STRING	Character string (max. 100 values)	**0 - 255**
AXIS	Axis addresses	**Axis names only**
FRAME	Geometric information	**As for axis paths**

from - to	REAL	INT	BOOL	CHAR	STRING
REAL		yes*	yes	yes**	-
INT	yes		yes	yes**	-
BOOL	yes	yes		yes	-
CHAR	yes	yes	yes**		yes
STRING	-	-	yes	yes***	

*   Value <>0 corresponds to TRUE, value ==0 corresponds to FALSE.
**   String length 0 => FALSE, otherwise TRUE
***   If only one character
It is not possible to convert from type AXIS and FRAME nor into type AXIS and FRAME.

Reaction:	Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy:	Modify the program section such that the value range is not exceeded, e.g. by a modified variable definition. Clear alarm with RESET key. Restart part program.

14097   **Channel %1 block %2 string cannot be converted to AXIS type**

Explanation:	%1 = Channel number %2 = Block number, label The called function AXNAME - conversion of the transferred parameters of the STRING type to an axis name (return value) of the AXIS type - has not found this axis identifier in the machine data.
Reaction:	Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy:	Check the transferred parameters (axis name) of the function AXNAME to determine whether a geometry, channel or machine axis of this name has been configured by means of the machine data: 10 000: AXCONF_MACHAX_NAME_TAB 20 070: AXCONF_GEOAX_NAME_TAB 20 080: AXCONF_CHANAX_NAME_TAB Select the transfer string in accordance with the axis name and change the axis name in the machine data if necessary. (If a change of name is to take place via the NC part program, this change must first be validated by means of a „Power On".) Clear alarm with RESET key. Restart part program.

14098   **Channel %1 block %2 conversion error: not a number**

Explanation:	%1 = Channel number %2 = Block number, label The string is not a valid INT or REAL number.
Reaction:	Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy:	Modify part program. If an input is concerned, it is possible to test whether the string represents a number by means of the predefined function ISNUMBER (with the same parameter). Clear alarm with RESET key. Restart part program.

14099   **Channel %1 block %2 result in string concatenation too long**

Explanation:	%1 = Channel number %2 = Block number, label The result of string chaining returns a string which is greater than the maximum string length laid down by the system.

emco

Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Teileprogramm anpassen.
      Adapt part program.
      With the function STRLEN, it is also possible to test the size of the sum string before performing the chaining operation.
      Clear alarm with RESET key. Restart part program.

14100   **Channel %1 block %2 orientation transformation not available**
Explanation: %1 = Channel number
      %2 = Block number, label
      Four transformation groupings (transformation types) can be set for each channel via machine data. If a transformation grouping is addressed by means of the keyword **TRAORI(n)** (n ... number of transformation grouping) but for which the machine data have no default values, then an alarm message is issued.
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Press the NC Stop key and select the function „Correction block" with the softkey PROGRAM CORRECT. The correction pointer positions on the incorrect block.
      • Check the number of the transformation grouping when calling the part program with the keyword **TRAORI(n)** (n ... number of the transformation grouping).
      • Enter the machine data for this transformation grouping and then activate by "Power On".
      Clear alarm with RESET key. Restart part program.

14115   **Channel %1 block %2 illegal definition of part surface**
Explanation: %1 = Channel number
      %2 = Block number, label
      The surface normal vectors programmed at the beginning of block and at the end of block point in opposite directions.
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Modify part program
      Clear alarm with RESET key. Restart part program.

14130   **Channel %1 block %2 too many initialization values given**
Explanation: %1 = Channel number
      %2 = Block number, label
      On assigning an array by means of SET, more initialization values than existing array elements have been specified in the program run.
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Reduce the number of initialization values.
      Clear alarm with RESET key. Restart part program.

14150   **Channel %1 block %2 illegal tool carrier number programmed or declared (MD)**
Explanation: %1 = Channel number
      %2 = Block number, label
      A toolholder number was programmed which is negative or greater than the machine data MC_MM_NUM_TOOL_CARRIER.
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable
Remedy: Program valid toolholder number or adapt machine data MC_MM_NUM_TOOL_CARRIER.
      Mit Reset-Taste Alarm löschen.

14200   **Channel %1 block %2 polar radius negative**
Explanation: %1 = Channel number
      %2 = Block number, label
      In the endpoint specification of a traversing block with G00, G01, G02 or G03 in polar coordinates, the polar radius entered for the keyword **RP=...** is negative.
      Definition of terms:
      • **Specification of end of block point** with polar angle and polar radius, referring to the current pole (preparatory functions: G00/G01/G02/G03).
      • **New definition of the pole** with polar angle and pole radius, referring to the reference point selected with the G function.
      G110 ... last programmed point in the plane
      G111 ... zero point in the actual WCS
      G112 ... last pole
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Correct NC part program - permissible inputs for the pole radius are only positive absolute values that specify the distance between the current pole and the block end point (the direction is defined by the polar angle **AP=...**).
      Clear alarm with RESET key. Restart part program.

14210   **Channel %1 block %2 polar radius too large**
Explanation: %1 = Channel number
      %2 = Block number, label
      In specifying the endpoints in a traversing block with G00, G01, G02 or G03 in polar coordinates, the value range of the polar angle programmed under the keyword **AP=...** has been exceeded. It covers the range from -360 to +360 degrees with a resolution of 0.0 01 degrees.
      Definition of terms:
      • **Specification of end of block point** with polar angle and polar radius, referring to the current pole (preparatory functions: G00/G01/G02/G03).

• **New definition of the pole** with polar angle and pole radius, referring to the reference point selected with the G function.
G110 ... referred to the last programmed point in the plane
G111 ... referred to the zero point of the current workpiece coordinate system (WCS)
G112 ... referred to the last pole

Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Correct NC part program. The permissible input range for the polar angle is between the values -360 degrees and +360 degrees with a resolution of 0.001 degrees.
              Clear alarm with RESET key. Restart part program.

14250   **Channel %1 block %2 pole radius negative**
Explanation: %1 = Channel number
              %2 = Block number, label
              in redefining the pole with G110, G111 or G112 in polar coordinates, the pole radius specified under keyword **RP=...** is negative. Only positive absolute values are permitted.
              Definition of terms:
              • **Specification of end of block point** with polar angle and polar radius, referring to the current pole (preparatory functions: G00/G01/G02/G03).
              • **New definition of the pole** with polar angle and pole radius, referring to the reference point selected with the G condition.
              G110 ... last programmed point in the plane
              G111 ... zero point of the current workpiece coordinate system (WCS)
              G112 ... last pole
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Correct the NC part program. Permissible inputs for the pole radius are only positive, absolute values that specify the distance between the reference point and the new pole (the direction is defined with the pole angle **AP=...**).
              Clear alarm with RESET key. Restart part program.

14260   **Channel %1 block %2 pole angle too large**
Explanation: %1 = Channel number
              %2 = Block number, label
              In redefining the pole with G110, G111 or G112 in polar coordinates, the value range of the pole angle specified under keyword **AP=...** has been exceeded. It covers the range from -360 to +360 degrees with a resolution of 0.001 degrees.
              Definition of terms:
              • **Specification of end of block point** with pole angle and pole radius, referring to the current pole (preparatory functions: G00/G01/G02/G03).
              • **New definition of the pole** with pole angle and pole radius, referring tothe reference point selected with the G function.
              G110 ... last programmed point in the plane
              G111 ... zero point of the current workpiece coordinate system (WCS)
              G112 ... last pole
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Correct NC part program. The permissible input range for the polar angle is between the values -360 degrees and +360 degrees with a resolution of 0.001 degrees.
              Clear alarm with RESET key. Restart part program.

14270   **Channel %1 block %2 pole programmed incorrectly**
Explanation: %1 = Channel number
              %2 = Block number, label
              When defining the pole, an axis was programmed that does not belong to the selected processing level. Programming in polar coordinates always refers to the plane activated with G17 to G19. This also applies to the definition of a new pole with G110, G111 or G112.
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Correct the NC part program. Only the two geometry axes may be programmed that establish the current machining plane.
              Clear alarm with RESET key. Restart part program.

14280   **Channel %1 block %2 polar coordinates programmed incorrectly**
Explanation: %1 = Channel number
              %2 = Block number, label
              The end point of the displayed block has been programmed both in the polar coordinate system (with AP=..., RP=...) and in the Cartesian coordinate system (axis addresses X, Y,...)..
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: NCCorrect the NC part program - the axis motion may be specified in **one coordinate system only**.
              Clear alarm with RESET key. Restart part program.

14300   **Channel %1 block %2 overlaid handwheel motion activated incorrectly**
Explanation: %1 = Channel number
              %2 = Block number, label
              Handwheel override has been called up incorrectly:
              1. For positioning axes:
              - Handwheel override programmed for indexing axes,
              - No position programmed,
              - FA and FDA programmed for the same axis in the block.
              2. For contouring axes:

	- No position programmed,

- No position programmed,
- G60 not active,
- 1st G group incorrect (only G01 to CIP)

Reaction:	Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy:	Modify part program.
	Clear alarm with RESET key. Restart part program.

14310   **Handwheel %1 configuration not correct or inactive**

Explanation: %1 = handwheel number
• The inputs are using a drive with a drive number that does not exist or
• an inactive drive for assignment of the handwheel (ENC_HANDWHEEL_MODULE_NR), or
• an axis is using a measuring circuit which does not exist for the drive hardware.

Reaction:	Alarm display. Interface signals are set. NC Start disable
Remedy:	Check input configuration (machine data) and/or drive hardware.
	Runup is interrupted.
	Switch control OFF - ON.

14400   **Channel %1 block %2 tool radius compensation active at transformation switchover**

Explanation: %1 = Channel number
%2 = Block number, label
A change of transformation is not allowed when tool radius compensation is active.

Reaction: Perform tool radius compensation in the NC part program with **G40** (in a block with G00 or G01) before performing a transformation change.
Clear alarm with RESET key. Restart part program.

14401   **Channel %1 block %2 transformation not available**

Explanation: %1 = Channel number
%2 = Block number, label
The required transformation is not available.
Example:
This was programmed: N220 TRAORI(3); 5-axis transform. no. 3 ON but only transformation 1 and 2 exist

Reaction:	Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy:	• Modify part program, program defined transformations only.
	• Check MD 24100 TRAFO_TYPE_n (assigns the transformation to part program instructions).
	Clear alarm with RESET key. Restart part program.

14403   **Channel %1 block %2 preparation might not be synchronized with interpolation**

Explanation: %1 = Channel number
%2 = Block number, label
Positioning axis runs cannot be accurately calculated beforehand. Consequently, the position in the MCS is not known exactly. It might therefore be possible that a change in the multiple significance of the transformation has been performed in the main run although no provision was made for this in the preprocessing run.

Reaction:	Alarm display.
Remedy:	Modify part program. Synchronize preprocessing run and main run.
	Clear alarm with the Cancel key. No further operator action necessary.

14404   **Channel %1 block %2 invalid argument in selection of transformation**

Explanation: %1 = Channel number
%2 = Block number, label
Error has occurred when selecting transformation.
Possible causes of error:
• An axis traversed by the transformation has not been enabled: · is being used by another channel (-> enable)
• is in spindle mode (-> enable with SPOS)
• is in POSA mode (-> enable with WAITP)
• is competing Pos axis (enable with -> WAITP)
• Parameterization via machine data has an error
• Axis or geometry axis assignment to the transformation has an error,
• Machine data has an error (-> modify machine data, cold restart)
Note: Any axes that have not been enabled might be signaled via EXINAL_ILLEGAL_AXIS = 14092 or BSAL_SYSERRCHAN_RESET = 1011 instead of EXINAL_TRANSFORM_PARAMETER = 14404.
Transformation-dependent error causes can be in:
TRAORI: -TRANSMIT:
• The current machine axis position is unsuitable for selection (e.g. selection in the pole) (-> change position slightly)
• Parameterization via machine data has an error
• Special requirement with respect to the machine axis has not been satisfied (e.g. rotary axis is not a modulo axis) (-> modify machine data, cold restart)
TRACYL:
• The programmed parameter is not allowed when transformation is selected.
TRAANG:

emco

• The programmed parameter is not allowed when transformation is selected.
• Parameterization via machine data has an error
• Parameter has an error (e.g. TRAANG: unfavorable angular value) (-> modify machine data, cold restart)

Reaction:	Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy:	Modify part program or machine data.
	Clear alarm with RESET key. Restart part program.

14411   **Channel %1 block %2 tool radius compensation active at change of geoaxis**
Explanation: %1 = Channel number
%2 = Block number, label
It is not permissible to change the assignment of geometry axes to channel axes when tool radius compensation is active.
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Modify part program.
Clear alarm with RESET key. Restart part program.

14412:   **Channel %1 block %2 transformation active at change of geoaxis**
Explanation: %1 = Channel number
%2 = Block number, label
It is not permissible to change the assignment of geometry axes to channel axes when transformation is active.
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Modify part program.
Clear alarm with RESET key. Restart part program.

14413   **Channel %1 block %2 fine tool correction: changeover geometry / channel**
Explanation: %1 = Channel number
%2 = Block number, label
It is not permissible to change the assignment of geometry axes to channel axes during active tool fine compensation.
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable
Remedy: Modify part program
Clear alarm with RESET key.

14414   **Channel %1 block %2 function GEOAX: incorrect call**
Explanation: %1 = Channel number
%2 = Block number, label
The parameters for the GEOAX(...) call are incorrect.
Possible causes are:
- Uneven number of parameters.
- More than 6 parameters were specified.
- A geometry axis number was programmed which was smaller than 0 or greater than 3.
- A geometry number was programmed more than once.
- An axis identifier was programmed more than once.
- An attempt was made to assign a channel axis to a geometry axis which has the same name as one of the channel axes.
- An attempt was made to remove a geometry axis from the geometry axis grouping and the geometry axis has the same name as one of the channel axes.
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable
Remedy: Modify part program or correction block
Cancel alarm with the Cancel key. No further operator action necessary.

14420   **Channel %1 block %2 index axis %3 frame not allowed**
Explanation: %1 = Channel number
%2 = Block number, label
%3 = axis
The axis is to be traversed as an indexing axis, but a frame is active. This is not allowed by machine data FRAME_OR_CORRPOS_NOTALLOWED.
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable
Remedy: Modify part program, change machine data CORR_FOR_AXIS_NOT_ALLOWED
Clear alarm with RESET key. Restart part program.

14500   **Channel %1 block %2 illegal DEF or PROC statement within part program**
Explanation: %1 = Channel number
%2 = Block number, label
NC part programs with high-level language elements are divided into a preceding **definition part** followed by a **program part.** The transition is not marked specifically; a definition statement is not allowed to follow the first program command.
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Put definition and PROFC statements at the beginning of the program.
Clear alarm with RESET key. Restart part program.

14510   **Channel %1 block %2 PROC statement missing on subroutine call**
Explanation: %1 = Channel number
%2 = Block number, label
subroutine calls with parameter transfer („call-by-value" or „call-by-reference") the called subroutine must begin with a PROC statement.

emco

Reaction:	Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy:	Define the subroutine in accordance with the type used.

1.. Conventional subroutine structure (without parameter transfer):
% SPF 123456
:
M17
2.. Subroutine structure with keyword and subroutine name (without parameter transfer):
PROC UPNAME
:
M17
ENDPROC
3. Subroutine structure with keyword and subroutine name (with parameter transfer "call-by-value"):
PROC UPNAME (VARNAME1, VARNAME2, ...)
:
M17
ENDPROC
4. Subroutine structure with keyword and subroutine name (with parameter transfer "call-by-reference"):
PROC UPNAME (Typ1 VARNAME1, Typ2 VARNAME2, ...)
:
M17
ENDPROC
Clear alarm with RESET key. Restart part program.

14520   **Channel %1 block %2 illegal PROC statement in data definition section**

Explanation: %1 = Channel number
%2 = Block number, label
The PROC statement may only be programmed at the beginning of the subroutine.

Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Modify NC part program appropriately.
Clear alarm with RESET key. Restart part program.

14530   **Channel %1 block %2 EXTERN and PROC statement do not correspond**

Explanation: %1 = Channel number
%2 = Block number, label
Subroutines with parameter transfer must be known **before** they are called in the program. If the subroutines are always available (fixed cycles) the control establishes the call interfaces at the time of system power-up. Otherwise an EXTERN statement must be programmed in the calling program.
Example:
N123 **EXTERN** UPNAME (TYPE1, TYPE2, TYPE3, ...)
The type of the variable must definitely correspond to the type given in the definition (PROC statements) or it must be compatible with it. The name can be different.

Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Check the variable types in the EXTERN and the PROC statements for correspondence and correct.
Clear alarm with RESET key. Restart part program.

14610   **Channel %1 block %2 compensation block not possible**

Explanation: %1 = Channel number
%2 = Block number, label
An alarm was output which could be eliminated basically via program correction. Since the error occurred in a program which is processed from external, a compensation block/program correction is not possible.

Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: - Abort program with reset.
- Correct program on MMC or PC.
- Restart reloading (possibly with block search and interrupt location).
Clear alarm with RESET key. Restart part program.

14660   **Channel %1 block %2 SETINT instruction uses with invalid input to trigger ASUP**

Explanation: %1 = Channel number
%2 = Block number, label
Asynchronous subroutines are subroutines that are executed following a hardware input (interrupt routine started by a rapid NCK input). The number of the NCK input must be between 1 and 8. It is provided with the keyword PRIO = ... with a priority of 1 - 128 (1 is the highest priority) in the SETINT statement.
Example:
If NCK input 5 changes to „1" the subroutine LIFT_Z should be started with the highest priority.
N100 SETINT (5) PRIO = 1 ABHEB_Z

Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
REMEDY: Program the NCK input of the SETINT statement with a value of not less than 1 or greater than 128.
Clear alarm with RESET key. Restart part program.

14750   **Channel %1 block %2 too many auxiliary functions programmed**

Explanation: %1 = Channel number
%2 = Block number, label
More than 10 auxiliary functions have been programmed in an NC block.

Reaction: Alarm display. Interface signals are set. Correction block is reorganized.
Remedy: Check whether all auxiliary functions are necessary in one block - modal functions need not be repeated. Create

emco

separate auxiliary function block or divide the auxiliary functions over several blocks.
Clear alarm with RESET key. Restart part program.

14760 **Channel %1 block %2 auxiliary function of a group programmed repeatedly**
Explanation: %1 = Channel number
%2 = Block number, label
The M and H functions can be divided up as required over machine data in groups in any variation. Auxiliary functions are thus put into groups that mutually preclude several individual functions of one group. Within one group only one auxiliary function is advisable and permissible.
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Program only one auxiliary function per auxiliary function group (group allocations: refer to the machine manufacturer's programming guide).
Clear alarm with RESET key. Restart part program.

14770 **Channel %1 block %2 auxiliary function programmed incorrectly**
Explanation: %1 = Channel number
%2 = Block number, label
The permissible number of programmed auxiliary functions per NC block has been exceeded or more than one auxiliary function of the same auxiliary function group has been programmed (M and S function). In the user-defined auxiliary functions, the maximum number of auxiliary functions per group in the NCK system settings has been defined for all auxiliary functions by means of the machine data 11100 **AUXFU_MAXNUM_GROUP_ASSIGN** (default: 1).
For each user-defined auxiliary function to be assigned to a group, the assignment is effected through 4 channel-specific machine data.
22010 **AUXFU_ASSIGN_TYPE: type of auxiliary function,** e.g. M
22000 **AUXFU_ASSIGN_GROUP:** required **group**
22020 **AUXFU_ASSIGN_EXTENSION:** any required **extension**
22030 **AUXFU_ASSIGN_VALUE: function value**
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Correct the part program - max. 16 auxiliary functions, max. 5 M functions per NC block, max. 1 auxiliary function per group.
Clear alarm with RESET key. Restart part program.

14820 **Channel %1 block %2 negative value for maximum spindle speed programmed with constant cutting speed**
Explanation: %1 = Channel number
%2 = Block number, label
For the function "Constant cutting speed G96" a maximum spindle speed can be programmed with the keyword **LIMS=...**. The values are in the range 0.1 -999 999.9 [rev/min].
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Program the maximum spindle speed for the constant cutting speed within the limits given above. The keyword LIMS is modal and can either be placed in front of or within the block that selects the constant cutting speed.
Clear alarm with RESET key. Restart part program.

14830 **Channel %1 block %2 wrong feed type selected**
Explanation: %1 = Channel number
%2 = Block number, label
Im G97 has been programmed in the displayed block although G96 was not (or G97 already) active previously.
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Remove G97 from the displayed block and program the correct feed type (G93, G94, G95 or G96) for the machining section which follows.
Clear alarm with RESET key. Restart part program.

14840 **Channel %1 block %2 value for constant cutting speed out of range**
Explanation: %1 = Channel number
%2 = Block number, label
The programmed cutting speed is not within the input range
Input range metric: 0.01 to 9 999.99 [m/min] Input range inch: 0.1 to 99 999.99 [inch/min]
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Program cutting speed under address S within the permissible range of values.
Clear alarm with RESET key. Restart part program.

14900 **Channel %1 block %2 use either center point or end point programming**
Explanation: %1 = Channel number
%2 = Block number, label
When programming a circle by means of the opening angle, the circle center point was programmed together with the circle end point. This is too much information for the circle. Only one of the two points is allowed.
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Select the programming variant guaranteeing that the dimensions are definitely taken over from the workpiece drawing (avoidance of calculation errors).
Clear alarm with RESET key. Restart part program.

14910 **Channel %1 block %2 invalid angle of aperture for programmed circle**
Explanation: %1 = Channel number
%2 = Block number, label
When programming a circle by means of the opening angle, a negative opening angle or an opening angle greater than or equal to 360 degrees has been programmed.
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.

emco

Remedy:	Program opening angle within the allowed range of values between 0.0001 and 359.9999 [degrees]. Clear alarm with RESET key. Restart part program.

14920 Channel %1 block %2 intermediate point of circle incorrect
Explanation: %1 = Channel number
%2 = Block number, label
When programming a circle by means of an intermediate point (CIP) all 3 points (initial, end and intermediate points) are on a straight line and the intermediate point (programmed by means of interpolation parameters I, J, K) is not located between the initial and end points. If the circle is the component of a helix, the specified number of turns (keyword TURN=...) determines further block processing:
• TURN>0: alarm display because the circle radius is infinitely great.
• TURN=0 and CIP specified between initial and end points. A straight line is generated between the initial and end points (without alarm message).
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Locate the position of the intermediate point with the parameters I, J and K in such a way that it actually is located between the initial and end points of the circle or do not make use of this type of circle programming and instead program the circle with radius or opening angle or center point parameters.
Clear alarm with RESET key. Restart part program.

15010 Channel %1 block %2 channel-sync instruction using illegal mark
Explanation: %1 = Channel number
%2 = Block number, label
A WAITM/WAITMC/SETM/CLEARM instruction was programmed with a marker number
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Correct the instruction accordingly.
Clear alarm with RESET key. Restart part program.

15180 Channel %1 block %2 program %3 cannot be executed as INI file
Explanation: %1 = Channel number
%2 = Block number, label
%3 = string
Errors occurred when reading in as INI file. The error message which is then displayed refers to the program specified here.
Reaction: Alarm display.
Remedy: Correct the part program.
Clear alarm with the Cancel key. No further operator action necessary.

15185 Channel %1 %2 errors in INI file
Explanation: %1 = Channel number
%2 = Number of detected errors
An error was found when processing an INI file
Reaction: Alarm display. Interface signals are set. NC Start disable.
Remedy: Correct the INI file or correct the MD and create a new INI file (via „Upload").
Switch control OFF-ON.

15300 Channel %1 block %2 invalid number-of-passed blocks during block search
Explanation: %1 = Channel number
%2 = Block number, label
In the function „Block search with calculation" a negative number of passes has been entered in column P (number of passes). The permissible range of values is P 1 - P 9 999.
Reaction: Alarm display.
Remedy: Enter only positive number of passes within the range of values.
Clear alarm with Cancel key. No further operator action necessary.

15310 Channel %1 block %2 file requested during block search is not loaded
Explanation: %1 = Channel number
%2 = Block number, label
During block search, a target has been specified with a program that has not been loaded
Reaction: Alarm display.
Remedy: Correct the specified search target accordingly or retroload the file
Clear alarm with the Cancel key. No further operator action necessary.

15320 Channel %1 block %2 invalid block search command
Explanation: %1 = Channel number
%2 = Block number, label
The block search command (type of search target) is smaller than 1 or greater than 5. It is entered in column **type** of the block search window. The following block search orders are allowed.
Type Meaning
1 Search for block number
2 Search for label
3 Search for string
4 Search for program name
5 Search for line number in a file
Reaction: Alarm display.
Remedy: Modify the block search command.
Clear alarm with the Cancel key. No further operator action necessary.

15330   **Channel %1 block %2 invalid block number as target of block search**
Explanation: %1 = Channel number
                          %2 = Block number, label
                          Syntax error! Positive integers are allowed as block numbers. Block numbers must be preceded by „:" and subblocks
                          by an „N".
Reaction: Alarm display.
Remedy: Repeat the input with corrected block number.
                          Clear alarm with the Cancel key. No further operator action necessary.

15340   **Channel %1 block %2 invalid label as target of block search**
Explanation: %1 = Channel number
                          %2 = Block number, label
                          Syntax error! A label must have at least 2 but no more than 32 characters, and the first two characters must be
                          alphabetic or underscore characters. Labels must be concluded with a colon.
Reaction: Alarm display.
Remedy: Repeat the input with corrected label.
                          Clear alarm with the Cancel key. No further operator action necessary.

15350   **Channel %1 block %2 target of block search not found**
Explanation: %1 = Channel number
                          %2 = Block number, label
                          The specified program has been searched to the end of the program without the selected search target having been
                          found.
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Check the part program, change the block search (typing error in the part program) and restart the search.
                          Clear alarm with RESET key. Restart part program.

15360   **Channel %1 invalid target of block search (syntax error)**
Explanation: %1 = Channel number
                          The specified search target (block number, label or string) is not allowed in block search.
Reaction: Alarm display.
Remedy: Correct object of block search.
                          Clear alarm with the Cancel key. No further operator action necessary.

15370   **Channel %1 target of block search not found**
Explanation: %1 = Channel number
                          In a block search, an impermissible search target has been specified (e.g. negative block number).
Reaction: Alarm display.
Remedy: Check the specified block number, label or character string. Repeat entry with correct search target.
                          Clear alarm with the Cancel key. No further operator action necessary.

15400   **Channel %1 block %2 selected initial ini file does not exist**
Explanation: %1 = Channel number
                          %2 = Block number, label
                          The operator has selected an INI block for a read, write or execution function which:
                          1. Does not exist in the NCK range or
                          2. Does not have the necessary protection level required for performing the function
Reaction: Alarm display.
Remedy: Check whether the selected INI block is contained in the file system of the NCK. The present protection level must
                          be selected to be at least equal to (or greater than) the protection level that has been defined for the read, write or
                          execution function at the time of creating the file.
                          Clear alarm with RESET key. Restart part program.

15410   **Channel %1 block %2 initialization file contains invalid M function**
Explanation: %1 = Channel number
                          %2 = Block number, label
                          The only M function allowed in an Init block is the M02, M17 or M30 end-of-program function.
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Remove all M functions from the Init block except for the end identifier. An Init block may contain value assignments
                          only (and global data definitions if they are not defined again in a program that can be executed later) but no motion
                          or synchronous actions.
                          Clear alarm with RESET key. Restart part program.

15420   **Channel %1 block %2 instruction not accepted in current mode**
Explanation: %1 = Channel number
                          %2 = Block number, label
                          In executing an Init block, the interpreter encountered an impermissible statement (e.g. a traversing statement).
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Remove all motion actions and auxiliary functions from the Init block except for the end identifier. An Init block may
                          contain value assignments only (and global data definitions if they are not defined again in a program that can be
                          executed later) but no motion or synchronous actions.
                          Clear alarm with RESET key. Restart part program.

15460   **Channel %1 block %2 syntax conflict with modal G functions**
Explanation: %1 = Channel number
                          %2 = Block number, label
                          The addresses programmed in the block are not compatible with the modal syntax-determining G function.

EMCO

	Example:
	N100 G01 ... I .. J.. K.. LF
Reaction:	Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy:	Correct the displayed block and ensure that the G functions and addresses in the block are in agreement.
	Clear alarm with RESET key. Restart part program.

15800 — **Channel %1 block %2 wrong starting condition for CONTPRON**

Explanation:
%1 = Channel number
%2 = Block number, label
The start conditions for contour preprocessing (keyword CONTPRON) are not correct:
• G40 (deselection of the tool radius compensation) is not active
• Spline or polynomial interpolation has been selected

Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Modify part program. Deselect spline of polynomial interpolation and/or tool radius compensation with G40.
Clear alarm with RESET key. Restart part program.

15810 — **Channel %1 block %2 wrong array dimension for CONTPRON**

Explanation:
%1 = Channel number
%2 = Block number, label
The number of columns in a contour table is a fixed quantity. The value required here must be taken from the relevant technology programming guide.

Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Correct the array definition for the contour table.
The number of rows is freely definable and corresponds to the number of contour elements (circles, straight lines). The number of columns is fixed (release 6/94: column number = 11).
Example: N100 DEF REAL KONTAB_1 [30, 11]
Clear alarm with RESET key. Restart part program.

15900 — **Channel %1 block %2 touch probe not available**
15910 — **Channel %1 block %2 touch probe not available**

Explanation:
%1 = Channel number
%2 = Block number, label
Alarm no.: 15 900 ... Measure with deletion of distance-to-go
Alarm no.: 15 910 ... Measure without deletion of distance-to-go
In the part program, an illegal probe has been programmed with the command MEAS (measure with deletion of distance-to-go) or MEAW (measure without distance-to-go). The probe numbers
0 ... no probe
1 ... probe 1
2 ... probe 2
are allowed, whether the probe is actually connected or not.
Example:
N10 MEAS=2 G01 X100 Y200 Z300 F1000; Probe 2 with deletion of distance-to-go

Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Include a probe number within the limits given above in the keyword MEAS=... or MEAW=... . This must correspond to the hardware connection of the probe.
Clear alarm with RESET key. Restart part program.

15950 — **Channel %1 block %2 no traverse motion programmed**
15960 — **Channel %1 block %2 no traverse motion programmed**

Explanation:
%1 = Channel number
%2 = Block number, label
Alarm no.: 15 950 ... Measure with deletion of distance-to-go
Alarm no.: 15 960 ... Measure without deletion of distance-to-go
In the part program, no axis or a traversing path of zero has been programmed with the command MEAS (measure with deletion of distance-to-go) or MEAW (measure without deletion of distance-to-go).

Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Correct the part program and add the axis address or the traversing path to the measurements block.
Clear alarm with RESET key. Restart part program.

16000 — **Channel %1 block %2 invalid value for lifting direction**

Explanation:
%1 = Channel number
%2 = Block number, label
In „Rapid lift from the contour" (keyword: LIFTFAST) a code value has been programmed for the direction of lift (keyword: ALF=...) that is outside of the permissible range (permitted range of values: 0 to 8).
With active cutter radius compensation:
Code numbers 2, 3 and 4 cannot be used in G41
Code numbers 6, 7 and 8 cannot be used in G42 because they code the direction to the contour.

Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Program the lifting direction under ALF=... within the permissible limits.
Clear alarm with RESET key. Restart part program.

16005 — **Channel %1 block %2 invalid value for lifting distance**

Explanation:
%1 = Channel number
%2 = Block number, label
Mistake in the programming: the value for the lifting path must not be negative.

Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable

emco

Remedy:	Modify part program.
	Clear alarm with RESET key.

16020   **Channel %1 repositioning in block %2 is not possible.**
Explanation: %1 = Channel number
%2 = Block number, label
Programming or operator action incorrect: A block is to be approached again for which there is no repositioning information (e.g. REPOS programmed but no REORG performed, REPOS with A spline or B spline).
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Change part program if necessary.
Clear alarm with RESET key. Restart part program.

16100   **Channel %1 block %2 spindle %3 not available in channel**
Explanation: %1 = Channel number
%2 = Block number, label
%3 = String
Mistake in programming: This channel does not recognize the spindle number.
The alarm can occur together with a dwell or SPI function.
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Check the **part program** to determine whether the programmed spindle number is correct and whether the program is run in the correct channel. Check MD 35000 **SPIND_ASSIGN_TO_MACHAX** for all machine axes to see whether one of them contains the programmed spindle number. This machine axis number must be entered in a channel axis of the channel-specific machine data 20070 **AXCONF_MACHAX_USED**.
Clear alarm with RESET key. Restart part program.

16110   **Channel %1 block %2 spindle %3 for dwell time not in speed control mode**
Explanation: %1 = Channel number
%2 = Block number, label
%3 = Axis, spindle
The spindle can be in the positioning mode, oscillating mode and control mode. With the M command M70 it can be changed from a spindle to an axis. The control mode is divided into the speed-controlled and position-controlled mode, and it is possible to alternate between these with the keywords SPCON and SPCOF.
Positioning mode: Position control (spindle position under SPOS/SPOSA)
Oscillating mode: Speed control (M41 - M45 or M40 and S...)
Control mode: Speed control (spindle speed under S..., M3/M4/M5) Position control (SPCON/SPCOF, spindle speed under S..., M3/M4/M5)
Axis mode: Position control (M70/M3, M4, M5, axis position under user-selectable axis name)
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Check part program for correct spindle number.
With M3, M4 or M5 put the required spindle into control mode before calling the dwell time.
Clear alarm with RESET key. Restart part program.

16120   **Channel %1 block %2 invalid index for online tool compensation**
Explanation: %1 = Channel number
%2 = Block number, label
Mistake in programming:
The 2nd parameter in the PUTFTOC command indicates for which tool parameter the value is to be corrected (1 - 3 tool lengths, 4 tool radius). The programmed value is beyond the permitted range. Permissible values are 1 - 4 if on-line tool radius compensation is allowed (see machine data ONLINE_CUTCOM_ENABLE), otherwise values 1 - 3.
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Modify part program: Length 1 - 3 or 4 permissible for radius
Clear alarm with RESET key. Restart part program.

16130   **Channel %1 block %2 instruction not allowed with active FTOCON**
Explanation: %1 = Channel number
%2 = Block number, label
Case 1:
Change of plane is not allowed if the modal G function FTOCON: „Tool fine compensation on" is active.
Case 2:
Transformation selection is allowed only for zero transformation or transformation inclined axis, Transmit or Tracyl if FTOCON is active.
Case 3:
Tool change is not allowed with M06 if FTOCON has been active since the last tool change.
Case 4:
Orientable tool holder is active.
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Modify part program. Deselect fine tool compensation with FTOCOF
Clear alarm with RESET key. Restart part program.

16140   **Channel %1 block %2 FTOCON not allowed**
Explanation: %1 = Channel number
%2 = Block number, label
The tool fine compensation (FTOC) is not compatible with the currently active transformation.

emco

Reaction:	Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy:	Modify part program. Deselect tool fine compensation with FTOCOF
	Clear alarm with RESET key. Restart part program.

16150   **Channel %1 block %2 invalid spindle no. with PUTFTOCF**

Explanation: %1 = Channel number
%2 = Block number, label
The spindle number programmed for PUTFTOC or PUTFTOCF is beyond the permitted range for the spindle numbers.

Reaction:	Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy:	Modify part program. Is the programmed spindle number available?
	Clear alarm with RESET key. Restart part program.

16410   **Channel %1 block %2 axis %3 is not a geometry axis**

Explanation: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number
A geometry axis has been programmed that cannot be imaged on any machine axis in the current transformation (possibly there is no transformation active at the moment).
Example:
Without transformation: Polar coordinate system with X, Z, and C axis
With transformation: Cartesian coordinate system with X, Y, and Z e.g. with TRANSMIT.

Reaction:	Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy:	Activate transformation type with TRAORI (n) or do not program geometry axes that do not participate in the transformation grouping.
	Clear alarm with RESET key. Restart part program.

16420   **Channel %1 block %2 axis %3 repeatedly programmed**

Explanation: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number
It is not allowed to program an axis more than once.

Reaction:	Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy:	Delete the axis addresses that have been programmed more than once.
	Clear alarm with RESET key. Restart part program.

16430   **Channel %1 block %2 geometry axis %3 cannot traverse as positioning axis in rotated coordinate system**

Explanation: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number
In the rotated coordinate system, traversing of a geometry axis as positioning axis (i.e. along its axis vector in the rotated coordinate system) would mean traversing of several machine axes. This is in conflict with the positioning axis concept, however, in which **one** axis interpolator runs in addition to the path interpolator!

Reaction:	Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy:	Traverse geometry axes as positioning axes only with rotation deactivated.
	Deactivate rotation: Keyword ROT without further specification of axis and angle.
	Exasmple: N100 ROT
	Clear alarm with RESET key. Restart part program.

16500   **Channel %1 block %2 chamfer or radius negative**

Explanation: %1 = Channel number
%2 = Block number, label
A negative chamfer or rounding has been programmed under the keywords CHF= ..., RND=... or RNDM=... .

Reaction:	Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy:	Values for chamfers, roundings and modal roundings must be programmed with positive values only.
	Clear alarm with RESET key. Restart part program.

16510   **Channel %1 block %2 facing axis is not defined**

Explanation: %1 = Channel number
%2 = Block number, label
Diameter programming has been activated with the keyword DIAMON although no facing axis has been programmed in this NC block. If the diameter axis is not a geometry axis, in the initial setting „DIAMON" the alarm appears as soon as the control is switched on.

Reaction:	Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy:	Activate the modal G function DIAMON only in NC blocks containing a facing axis or deactivate diameter program with DIAMOF.
	In machine data 20150 GCODE_RESET_VALUES[28] select „DIAMOF" for the initial setting.
	Clear alarm with RESET key. Restart part program.

16700   **Channel %1 block %2 axis %3 invalid feed type**

Explanation: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number
At a thread cutting operation the feed was programmed in a wrong unit.

1. **G33 (thread with constant lead)** and the feed have not been programmed with G94 or G95.
2. **G33 (thread with constant lead)** is active (modal) and G63 is programmed additionally in a following block conflict situation! -> (G63 is in the 2nd G group, G33, G331 and G332 are in the 1st G group).
3. **G331** or **G332 (rigid tapping)** and the feed have not been programmed with G94.

Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Use only the feed type G94 or G95 in the thread cutting functions.
After G33 and before G63, deselect the thread cutting function with G01.
Clear alarm with RESET key. Restart part program.

16710   **Channel %1 block %2 axis %3 master spindle not programmed**
Explanation: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number
A master spindle function has been programmed (G33, G331, G95, G96) but the speed or the direction of rotation of the master spindle is missing.
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Add S value or direction of rotation for the master spindle in the displayed block.
Clear alarm with RESET key. Restart part program.

16715   **Channel %1 block %2 axis %3 master spindle not in standstill**
Explanation: %1 = Channel number
%2 = Block number, label
%3 = Spindle number
In the applied function (G74, reference point approach), the spindle must be stationary.
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Program M5 or SPOS/SPOSA in front of the defective block in the part program.
Clear alarm with RESET key. Restart part program.

16720   **Channel %1 block %2 axis %3 thread lead is zero**
Explanation: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number
No lead was programmed in a thread block with **G33** (thread with constant lead) or **G331** (rigid tapping).
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: The thread lead must be programmed for the specified geometry axis under the associated interpolation parameters. X -> I, Y -> J, Z -> K
Clear alarm with RESET key. Restart part program.

16730   **Channel %1 block %2 axis %3 wrong parameter for thread cutting**
Explanation: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number
In **G33** (tapping with constant lead) the lead parameter was not assigned to the axis that determines the velocity. For longitudinal and face threads, the thread lead for the specified geometry axis must be programmed under the associated interpolation parameter. X -> I, Y -> J, Z -> K
For taper threads, the address **I, J, K** depends on the axis with the longer path (thread length). A 2nd lead for the other axis is, however, **not specified.**
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Assign lead parameters to the axis that determines the velocity.
Clear alarm with RESET key. Restart part program.

16740   **Channel %1 block %2 geometry axis must be programmed**
Explanation: %1 = Channel number
%2 = Block number, label
No geometry axis was programmed for tapping (G33) or for rigid tapping (G331, G332). The geometry axis is, however, essential if an interpolation parameter has been specified.
Example:
N100 G33 Z400 K2 ; thread lead 2mm, thread: end Z=400mm
N200 SPOS=0 ; position spindle in axis mode
N201 G90 G331 Z-50 K-2 ; tapping to Z=-50, counterclockwise
N202 G332 Z5 ; retraction, direction reversal automatic
N203 S500 M03 ; spindle again in spindle mode
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Specify geometry axis and corresponding interpolation parameters.
Clear alarm with RESET key. Restart part program.

16750   **Channel %1 block %2 axis %3 SPCON not programmed**
Explanation: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number
For the programmed function (rotary axis, positioning axis), the spindle must be in position control mode.
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Program position control of the spindle with SPCON in the previous block.
Clear alarm with RESET key. Restart part program.

emco

16751   **Channel %1 block %2 spindle/axis %3 SPCOF.**
Explanation: %1 = Channel number
                  %2 = Block number, label
                  %3 = Axis name, spindle number
                  For the programmed function, the spindle must be in the open-loop control mode. In the positioning or axis mode, the position control must not be deselected.
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Put the spindle into open-loop control mode in the preceding block. This can be done with M3, M4 or M5 for the relevant spindle.
                  Clear alarm with RESET key. Restart part program.

16755   **Channel %1 block %2 no wait needed**
Explanation: %1 = Channel number
                  %2 = Block number, label
                  No Stop is needed for the programmed function. A Stop is necessary after SPOSA or after M5 if the next block is to be applied only after the spindle has come to a stop.
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Do not write instruction.
                  Clear alarm with RESET key. Restart part program.

16760   **Channel %1 block %2 axis %3 S value missing**
Explanation: %1 = Channel number
                  %2 = Block number, label
                  %3 = Axis name, spindle number
                  No spindle speed has been given for rigid tapping (G331 or G332).
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Program the spindle speed under address S in [rpm] (in spite of axis mode); the direction of rotation is given by the sign of the spindle lead.
                  Positive thread lead: Rotational direction as M03
                  Negative thread lead: Rotational direction as M04
                  Clear alarm with RESET key. Restart part program.

16761   **Channel %1 block %2 axis/spindle %3 not programmable in channel**
Explanation: %1 = Channel number
                  %2 = Block number, label
                  %3 = Axis name, spindle number
                  Mistake in the programming: The axis / spindle can not be programmed in the channel at this time. This alarm can occur when the axis / spindle is being used by another channel or by the PLC.
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Modify part program, use „GET()".
                  Clear alarm with RESET key. Restart part program.

16762   **Channel %1 block %2 spindle %3 function of thread or drill is active**
Explanation: %1 = Channel number
                  %2 = Block number, label
                  %3 = Spindle number
                  Mistake in programming: The spindle function cannot be executed at the present time. This alarm occurs when the spindle (master spindle) is linked with the axes by an interpolation function.
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Modify part program. Deselect thread cutting or tapping.
                  Clear alarm with RESET key. Restart part program.

16763   **Channel %1 block %2 axis %3 programmed speed is illegal (zero or negative)**
Explanation: %1 = Channel number
                  %2 = Block number, label
                  %3 = Axis name, spindle number
                  A spindle speed (S value) was programmed with the value zero or with a negative value.
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable
Remedy: The programmed spindle speed (S value) must be positive. Depending on the application case, the value zero can be accepted (e.g. G25 S0).
                  Clear alarm with RESET key.

16770   **Channel %1 block %2 axis %3 encoder missing**
Explanation: %1 = Channel number
                  %2 = Block number, label
                  %3 = Axis name, spindle number
                  SPCON, SPOS or SPOSA has been programmed. These functions require at least one measuring system. According to MD: NUM_ENCS the machine axis/spindle has no measuring system.
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Retrofit a measuring system.
                  Clear alarm with RESET key. Restart part program.

16783   **Channel %1 block %2 slave axis/spindle %3 currently not available**

Explanation:	%1 = Channel number
	%2 = Block number, label
	%3 = Axis name, spindle number
	A coupling has been switched on in which the slave spindle/axis is currently not available. Possible causes are:
	• The spindle/axis is active in the other channel.
	• The spindle/axis has been operated from the PLC and has not yet been enabled.
Reaction:	Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Abhhilfe:	Put the master spindle/axis with spindle/axis exchange into the necessary channel or release by the PLC.
	Clear alarm with RESET key. Restart part program.

16785   **Channel %1 block %2 master and slave axis/spindle %3 are identical**

Explanation:	%1 = Channel number
	%2 = Block number, label
	%3 = Axis name, spindle number
	A coupling has been switched on in which the following spindle/axis is identical to the master spindle/axis
Reaction:	Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy:	• Configure link accordingly in MD (channel MD: COUPLE_AXIS_n)
	• or correct the part program.
	Clear alarm with RESET key. Restart part program.

16800   **Channel %1 block %2 traverse instruction DC/CDC for axis %3 not allowed**

Explanation:	%1 = Channel number
	%2 = Block number, label
	%3 = Axis name, spindle number
	The keyword DC (**D**irect **C**oordinate) can only be used for rotary axes. This causes approach of the programmed **absolute position** along the shortest path.
	Example:
	N100 C=DC(315)
Reaction:	Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy:	Replace the keyword DC in the displayed NC block by specifying **AC** (**A**bsolute **C**oordinate).
	If the alarm display is the result of an error in the axis definition, the axis can be declared as a rotary axis by means of the axis-specific MD 30 300 **IS_ROT_AX**.
	Corresponding machine data:
	MD 30 310: **ROT_IS_MODULO**
	MD 30 320: **DISPLAY_IS_MODULO**
	Clear alarm with RESET key. Restart part program.

16810   **Channel %1 block %2 traverse instruction ACP for axis %3 not allowed**

Explanation:	%1 = Channel number
	%2 = Block number, label
	%3 = Axis name, spindle number
	The keyword **ACP** (**A**bsolute **C**oordinate **P**ositive) is only allowed for "modulo axes". It causes approach of the programmed **absolute position** in the specified direction.
Reaction:	Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy:	In the displayed NC block, replace the keyword ACP by specifying **AC** (**A**bsolute **C**oordinate). If the alarm display is based on an incorrect axis definition, the axis with the axis-specific MD 30 300: **IS_ROT_AX** and MD 30 310: **ROT_IS_MODULO** can be declared a rotary axis with modulo change.
	Corresponding machine data:
	MD 30 320: **DISPLAY_IS_MODULO**
	Clear alarm with RESET key. Restart part program.

16820   **Channel %1 block %2 traverse instruction ACN for axis %3 not allowed**

Explanation:	%1 = Channel number
	%2 = Block number, label
	%3 = Axis name, spindle number
	The keyword **ACN** (**A**bsolute **C**oordinate **N**egative) is only allowed for "modulo axes". It causes approach of the programmed **absolute position** in the specified direction.
Reaction:	Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy:	In the displayed NC block, replace the keyword ACN by specifying **AC** (**A**bsolute **C**oordinate). If the alarm display is based on an incorrect axis definition, the axis with the axis-specific MD 30 300: **IS_ROT_AX** and MD 30 310: **ROT_IS_MODULO** can be declared a rotary axis with modulo change.
	Corresponding machine data: MD 30 320: **DISPLAY_IS_MODULO**
	Clear alarm with RESET key. Restart part program.

16830   **Channel %1 block %2 invalid position for axis/spindle %3 programmed**

Explanation:	%1 = Channel number
	%2 = Block number, label
	%3 = Axis name, spindle number
	A position beyond the range of 0 - 359.999 has been programmed for a modulo axis.
Reaction:	Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy:	Program position in the range 0 - 359.999.
	Clear alarm with RESET key. Restart part program.

16903   **Channel %1 program control: action %2 not allowed in current state**

emco

Explanation:	%1 = Channel number
	%2 = Action number/action name
	The relevant action cannot be processed now. This can occur, for instance, during read-in of machine data.
Reaction:	Alarm display
Remedy:	Wait until the procedure is terminated or abort with Reset and repeat the operation.
	Clear alarm with the Cancel key. No further operator action necessary.

16904   **Channel %1 program control: action %2 not allowed in current state**
Explanation:	%1 = Channel number
	%2 = Action number/action name
	The operation (program, JOG, block search, reference point, etc.) cannot be started or continued in the current status.
Reaction:	Alarm display
Remedy:	Check the program status and channel status.
	Clear alarm with the Cancel key. No further operator action necessary.

16905   **Channel %1 program control: action %2 not allowed**
Explanation:	%1 = Channel number
	%2 = Action number/action name
	Operation cannot be started or continued. A start is only accepted when an NCK function can be started.
	Example:
	A start is accepted in JOG mode when, for example, the function generator is active or a JOG movement has first been stopped with the Stop key.
Reaction:	Alarm display depending on MD 11411 ENABLE_ALARM_MASK
Remedy:	Check the program status and channel status.
	Clear alarm with the Cancel key. No further operator action necessary.

16906   **Channel %1 program control: action %2 is aborted because of an active alarm**
Explanation:	%1 = Channel number
	%2 = Action number/action name
	The action was aborted due to an alarm.
Reaction:	Alarm display
Remedy:	Remedy the error and acknowledge the alarm. Then repeat the operation.
	Clear alarm with the Cancel key. No further operator action necessary.

16907   **Channel %1 action %2 only possible in stop**
Explanation:	%1 = Channel number
	%2 = Action number/action name
	This action may only be performed in Stop state
Reaction:	Alarm display
Remedy:	Check the program status and channel status.
	Clear alarm with the Cancel key. No further operator action necessary.

16908   **Channel %1 action %2 only possible in reset or at the block end**
Explanation:	%1 = Channel number
	%2 = Action number/action name
	This action may only be performed in Reset state or at end of block.
Reaction:	Alarm display
Remedy:	Check the program status and channel status
	Clear alarm with the Cancel key. No further operator action necessary.

16909   **Channel %1 action %2 is not allowed in current mode**
Explanation:	%1 = Channel number
	%2 = Action number / action name
	You have to activate a different operating mode for the function to be activated.
Reaction:	Alarm display
Remedy:	Check operation and operating state.
	Clear alarm with the Cancel key. No further operator action necessary.

16911   **Channel %1 mode change is not allowed**
Explanation:	%1 = Channel number
	The change from overstoring in another operating mode is not allowed.
Reaction:	Alarm display
Remedy:	After overstoring is terminated, it is possible to change to another operating state again.
	Clear alarm with the Cancel key. No further operator action necessary.

16912   **Channel %1 program control: action %2 only possible in reset**
Explanation:	%1 = Channel number
	%2 = Action number / action name
	This action can only be performed in Reset state.
	Example:
	Program selection through MMC or channel communication (INIT) can only be performed in Reset state.
Reaction:	Alarm display
Remedy:	Reset or wait until processing is terminated.

emco

Clear alarm with the Cancel key. No further operator action necessary.

16913   **Mode group %1 channel %2 mode change: action %3 not allowed**
Explanation: %1 = Channel number
                  %2 = Mode group number
                  %3 = Action number / action name
                  The change to the desired mode is not permitted. The change can only take place in the Reset state.
                  Example:
                  Program processing is halted in AUTO mode by NC Stop. Then there is a mode change to JOG mode (program
                  status interrupted). From this operating mode it is only possible to change to AUTO mode and not to MDA mode!
Reaction: Alarm display
Remedy: Either activate the Reset key to reset program processing, or activate the mode in which the program was being
                  processed previously.   Clear alarm with the Cancel key. No further operator action necessary.

16914   **Mode group %1 channel %2 mode change: action %3 not allowed**
Explanation: %1 = Channel number
                  %2 = Mode group number
                  %3 = Action number / action name
                  Incorrect mode change, e.g.: Auto->MDAREF
Reaction: Alarm display
Remedy: Check operation or selected mode.
                  Clear alarm with the Cancel key. No further operator action necessary.

16915   **Channel %1 action %2 in the current block not allowed**
Explanation: %1 = Channel number
                  %2 = Action number / action name
                  If traversing blocks are interrupted by asynchronous subroutines, then it must be possible for the interrupted
                  program to continue (reorganization of block processing) after termination of the asynchronous subroutine.
                  The 2nd parameter describes which action wanted to interrupt block processing.
Reaction: Alarm display
Remedy: Let the program continue to a reorganized NC block or modify part program.
                  Clear alarm with the Cancel key. No further operator action necessary.

16916   **Channel %1 reposition: action %2 not allowed in the current state**
Explanation: %1 = Channel number
                  %2 = Action number / action name
                  Repositioning of block processing presently not possible.
                  In certain cases this can prevent a mode change from taking place.
                  The 2nd parameter describes which action should be used to perform repositioning.
Reaction: Alarm display
Remedy: Let the program continue to a repositioned NC block or modify part program.
                  Clear alarm with the Cancel key. No further operator action necessary.

16918   **Channel %1 for action %2 needs reset in all channels**
Explanation: %1 = Channel number
                  %2 = Action number / action name
                  All channels must be in the initial setting in order to carry out the action! (For example, for machine data loading)
Reaction: Alarm display
Remedy: Either wait until the channel status is aborted or press the Reset key.
                  Clear alarm with the Cancel key. No further operator action necessary.

16919   **Channel %1 action %2 is not allowed, because of an alarm**
Explanation: %1 = Channel number
                  %2 = Action number / action name
                  This action cannot be performed due to an alarm, or the channel is in Fail
Reaction: Alarm display
Remedy: Press RESET key
                  Clear alarm with the Cancel key. No further operator action necessary.

16920   **Channel %1 action %2 is already enabled**
Explanation: %1 = Channel number
                  %2 = Action number / action name
                  An identical action is still active.
Reaction: Alarm display
Remedy: Wait until the first procedure is terminated or abort with Reset and repeat the operation.
                  Clear alarm with Cancel key. No further operator action necessary.

16923   **Channel %1 program control: action %2 not allowed in the current state**
Explanation: %1 = Channel number
                  %2 = Action number / action name
                  The current processing cannot be stopped, due to an active preprocessing process. This applies to, for example,
                  loading machine data and block searches until the search object is found.
Reaction: Alarm display. Interface signals are set
Remedy: Abort by pressing Reset!
                  Clear alarm with Cancel key. No further operator action necessary.

16924   **Channel %1 caution: program test will of change the tool data**
Explanation: %1 = Channel number

emco

Tool management data is changed during program testing. It is not possible to automatically rectify the data after termination of the program testing. This error message prompts the user to make a backup copy of the data or to reimport the data after the operation is terminated.

Reaction: Alarm display
Remedy: Save tool data on MMC and reimport data after „ProgtestOff".
                 Clear alarm with Cancel key. No further operator action necessary.

16925   **Channel %1 program control: action %2 not allowed in the current state action %3 active**
Explanation: %1 = Channel number
                 %2 = Action number / action name
                 %3 = Action number / action name
                 The action has been refused since a mode or sub-mode change (change to automatic mode, MDA, JOG, overstoring, digitizing, etc.) is taking place.
                 Example: This alarm message is output if the Start key is pressed during a mode or sub-mode change from, for example, automatic to MDA, before the NCK has confirmed selection of the mode.

Reaction: Alarm display
Remedy: Repeat action. Clear alarm with Cancel key. No further operator action necessary.

16930   **Channel %1: Predecessor and current block %2 must be separated by an executable block**
Explanation: %1 = Channel number
                 %2 = Block number
                 The language functions WAITMC, SETM, CLEARM and MSG must be packed in separate NC blocks due to the language definition. To avoid velocity drops, these blocks are attached to the following NC block internally in the NCK (for WAITMC to the previous NC_block). For this reason, there must always be an executable block (no calculation block) between the NC blocks. An executable NC block includes always e.g. travel movements, a help function, Stopre, dwell time etc.

Reaction: Alarm display. Interface signals are set. Interpreter stop Correction block is reorganized
Remedy: Program an executable NC block between the previous and the current NC block.
                 Example:
                 N10 SETM
                 N15 STOPRE; insert executable NC block
                 N20 CLEARM
                 Clear alarm with NC Start. Restart part program.

17020   **Channel %1 block %2 1st array index out of range**
Explanation: %1 = Channel number
                 %2 = Block number, label
                 A read or write access has been programmed to an array variable with invalid 1st array index. The valid array indices must be contained within the defined array size and the absolute limits (0 - 32 766).

Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Correct the specification of array elements in the access instruction to match the defined size.
                 Clear alarm with RESET key. Restart part program.

17030   **Channel %1 block %2 2nd array index out of range**
Explanation: %1 = Channel number
                 %2 = Block number, label
                 A read or write access has been programmed to an array variable with invalid 2nd array index. The valid array indices must be contained within the defined array size and the absolute limits (0 - 32 766).

Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Correct the specification of array elements in the access instruction to match the defined size.
                 Clear alarm with RESET key. Restart part program.

17040   **Channel %1 block %2 illegal axis index**
Explanation: %1 = Channel number
                 %2 = Block number, label
                 A read or write access has been programmed to an axial variable in which the axis name cannot be unambiguously imaged on a machine axis.
                 Example:
                 Writing of an axial machine data.
                 $MA_... [X]= ... ; but geometry axis X cannot be imaged on a machine axis; because of a transformation

Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Deselect transformation before writing the axial data (keyword: TRAFOOF) or use the machine axis name as axis index.
                 Clear alarm with RESET key. Restart part program.

17050   **Channel %1 block %2 illegal value**
Explanation: %1 = Channel number
                 %2 = Block number, label
                 On accessing an individual frame element, a frame component other than TRANS, ROT, SCALE or MIRROR was addressed or the function CSCALE has been given a negative scale factor.
                 Example:
                 $P_UIFR[5] = CSCALE (X, -2.123)
                 The frame components are either selected by means of the keywords
                 TR for translation (TRANS, internal 0)
                 RT for rotation (ROT, internal 1)
                 SC for scaling and (SCALE, internal 3)

MI for mirroring (MIRROR, internal 4)
or they are specified directly as an integral value 0, 1, 3, 4.
Example:
Access to the rotation around the X axis of the currently settable frame.
R10=$P_UIFR[$AC_IFRNUM, X, RT] can also be programmed as:
R10=$P_UIFR[$AC_IFRNUM, X, 1]

Reaction:	Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy:	Address frame components only with the keywords provided; program the scale factor between the limits of 0.000 01 to 999.999 99. Clear alarm with RESET key. Restart part program.

17070   **Channel %1 block %2 data is write-protected**

Explanation:	%1 = Channel number %2 = Block number, label An attempt was made to write a write-protected variable (e.g. a system variable) or a machine data for which a higher protection level has been declared than the one currently active.
Reaction:	Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy:	Remove write access to write-protected system variables from the NC program. Increase the current protection level for writing the machine data. Clear alarm with RESET key. Restart part program.

17160   **Channel %1 block %2 tool is not selected**

Explanation:	%1 = Channel number %2 = Block number, label An attempt has been made to access the current tool offset data via the system variables: $P_AD [n] Contents of the parameter (n: 1 - 25) $P_TOOL Active D number (tool edge number) $P_TOOLL [n] Active tool length (n: 1 - 3) $P_TOOLR Active tool radius although no tool had been selected previously.
Reaction:	Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy:	Program or activate a tool offset in the NC program before using the system variables. Example: N100 G.. ... T5 D1 ... LF With the channel-specific machine data: MD 22 550:**TOOL_CHANGE_MODE** New tool offset for M function MD 22 560:**TOOL_CHANGE_M_CODE** M function with tool change It is established whether a tool offset is activated in the block with the T word or whether the new offset values are allowed for only when the M word for tool change occurs. Clear alarm with RESET key. Restart part program.

17170   **Channel %1 block %2 too many symbols defined**

Explanation:	%1 = Channel number %2 = Block number, label The predefined symbols could not be read in during power-up.
Reaction:	Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy:	Clear alarm with RESET key. Restart part program.

170180   **Channel %1 block %2 illegal D number**

Explanation:	%1 = Channel number %2 = Block number, label In the displayed block, access is made to a D number (tool edge number) that is not initialized and therefore is not available.
Reaction:	Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy:	Check tool call in the NC part program: • Correct tool edge number D.. programmed? If no tool edge number is specified, then D1 is automatically active. • Tool parameters P1 - P25 defined? The dimensions of the tool edge must have been entered previously either through the operator panel or through the V.24 interface. Description of the system variables $P_DP x [n, m] n ... Associated tool number T m ... Tool edge number D x ... Parameter number P Clear alarm with RESET key. Restart part program.

17190   **Channel %1 block %2 illegal T number**

Explanation:	%1 = Channel number %2 = Block number, label In the displayed block, access is made to a T number (tool number) that is not initialized and therefore not available.
Reaction:	Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy:	Check tool call in the NC part program: • Correct tool number T.. programmed? • Tool parameters P1 - P25 defined?

emco

The dimensions of the tool edge must have been previously entered either through the operator panel or through the V.24 interface.
Description of the system variables $P_DP x [n, m]
n ... Associated tool number T
m ... Tool edge number D
x ... Parameter number P
Clear alarm with RESET key. Restart part program.

17200   **Channel %1 block %2 cannot delete an active tool**
Explanation: %1 = Channel number
                %2 = Block number, label
                An attempt has been made to delete from the part program the tool data for a tool currently being processed. Tool data for tools involved in the current machining operation may not be deleted. This applies both for the tool preselected with T or that has been changed in place of another, and also for tools for which the constant grinding wheel peripheral speed or tool monitoring is active.
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Check access to tool offset memory by means of $TC_DP1[t,d] = 0 or deselect tool
                Clear alarm with RESET key. Restart part program.

17220   **Channel %1 block %2 tool not available**
Explanation: %1 = Channel number
                %2 = Block number, label
                If an attempt is made to access a tool via a T no. that has not (yet) been defined. For example, when tools are to be put into magazine locations by programming $TC_MPP6 = 'toolNo'. This is possible only when both the magazine location and the tool given by 'toolNo' have been defined.
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Correct the NC program. Clear alarm with RESET key. Restart part program.

17230   **Channel %1 block %2 Duplo no. already disposed**
Explanation: %1 = Channel number
                %2 = Block number, label
                If an attempt is made to write a tool Duplo number to the name of which another tool (another T number) already exists with the same Duplo number.
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Correct the NC program.
                Clear alarm with RESET key. Restart part program.

17240   **Channel %1 block %2 invalid definition of tool**
Explanation: %1 = Channel number
                %2 = Block number, label
                If an attempt is made to modify a tool data that would subsequently damage the data consistency or lead to a conflicting definition, this alarm will appear.
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Correct the NC program.
                Clear alarm with RESET key. Restart part program.

17250   **Channel %1 block %2 invalid definition of magazine**
Explanation: %1 = Channel number
                %2 = Block number, label
                If an attempt is made to modify a magazine data that would subsequently damage the data consistency or lead to a conflicting definition, this alarm will appear
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Correct the NC program.
                Clear alarm with RESET key. Restart part program.

17260   **Channel %1 block %2 invalid definition of magazine location**
Explanation: %1 = Channel number
                %2 = Block number, label
                If an attempt is made to modify a magazine location data that would subsequently damage the data consistency or lead to a conflicting definition, this alarm will appear
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Correct the NC program.
                Clear alarm with RESET key. Restart part program.

17270   **Channel %1 block %2 call-by-reference: illegal variable**
Explanation: %1 = Channel number
                %2 = Block number, label
                Machine data and system variables must not be transferred as call-by-reference parameters.
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Modify NC program: Assign the value of the machine data or of the system variable to a program-local variable and transfer this as parameter.
                Clear alarm with RESET key. Restart part program.

17500   **Channel %1 block %2 axis %3 is not an indexing axis**
Explanation: %1 = Channel number
                %2 = Block number, label
                %3 = Axis name, spindle number

emco

An indexing axis position has been programmed for an axis with the keywords CIC, CAC or CDC that has not been defined as indexing axis in the machine data.

Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Remove programming instruction for indexing axis positions (CIC, CAC, CDC) from the NC part program or declare the relevant axis to be an indexing axis.
Indexing axis declaration:
MD 30 500: **INDEX_AX_ASSIGN_POS_TAB**
(indexing axis assignment)
The axis becomes an indexing axis if an assignment to an indexing position table has been made in the specified MD.
Two tables are possible (input value 1 or 2).
MD 10 900: **INDEX_AX_LENGTH_POS_TAB_1**
MD 10 920: **INDEX_AX_LENGTH_POS_TAB_2**
(Number of positions for 1st/2nd indexing axis)
Standard value: 0 Maximum value: 60
MD 10 910: **INDEX_AX_POS_TAB_1 [n]**
MD 10 930: **INDEX_AX_POS_TAB_2 [n]**
(Positions of the 1st indexing axis) The absolute axis positions are entered. (The list length is defined via MD 10 900).
Clear alarm with RESET key. Restart part program.

17502 **Channel %1 block %2 indexing axis %3 with Hirth tooth system Stop delayed**
Explanation: %1 = Channel number
%2 = Block number, label
%3 = Axis name
For the indexing axis, the 'Hirth tooth system' function is activated and the override has been set to 0 or another stop condition (e.g. VDI interface signal) is active. Since it is possible to stop only on indexing axes, the next possible indexing position is approached. The alarm is displayed until this position is reached or the stop condition is deactivated.
Reaction: Alarm display.
Remedy: Wait until the next possible indexing position is reached or set override > 0 or deactivate another stop condition. Alarm display disappears with alarm cause. No further operation necessary.

17510 **Channel %1 block %2 invalid index for indexing axis %3**
Explanation: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number
The programmed index for the indexing axis is beyond the position table range.
Example:
Perform an absolute approach of the 56th position in the list allocated via the axis-specific machine data 30 500 **INDEX_AX_ASSIGN_POS_TAB** with the 1st positioning axis, the number of positions is e.g. only 40
(MD 10 900 **INDEX_AX_LENGTH_POS_TAB_1** = 40).
N100 G.. U=CAC (56)
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Program the indexing axis position in the NC part program in accordance with the length of the current position table, or add the required value to the position table and adjust the length of the list

17600 **Channel %1 block %2 preset on transformed axis %3 not possible**
Explanation: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number
The displayed axis is involved in the current transformation. This means that is it not possible to set the actual value memory (preset) for this axis.
Example:
The machine axis A should be set to the new actual value A 100 at the absolute position A 300.
:
N100 G90 G00 A=300
N101 PRESETON A=100
:
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Avoid preset actual value memory for axes, which are participating in a transformation, or deselect the transformation with the keyword **TRAFOOF**.
Clear alarm with RESET key. Restart part program.

17610 **Channel %1 block %2 positioning axis %3 cannot participate in transformation**
Explanation: %1 = Channel number
%2 = Block number, label
%3 = Axis name, spindle number
The axis addressed with the keyword POS or POSA is involved in the active transformation. Therefore, it cannot be traversed as a positioning axis.
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Remove the POS or POSA instruction from the part program block or previously deselect transformation with TRAFOOF.
Clear alarm with RESET key. Restart part program.

17620 **Channel %1 block %2 fixpoint cannot be approached for transformed axis %3**
Explanation: %1 = Channel number
%2 = Block number, label

emco

%3 = Axis name, spindle number
In the displayed block, an axis is programmed for the fixed point approach (G75) that is involved in the active transformation. Fixed point approach is not performed with this axis

Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
                 Remove G75 instruction from the part program block or previously deselect transformation with TRAFOOF
                 Clear alarm with RESET key. Restart part program.

17630   **Channel %1 block %2 referencing not possible for transformed axis %3**
Explanation: %1 = Channel number
                 %2 = Block number, label
                 %3 = Axis name, spindle number
                 In the displayed block, an axis is programmed for reference point approach (G74) that is involved in the active transformation. Reference point approach is not performed with this axis!
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Remove G74 instruction, or the machine axes involved in transformation, from the part program block or previously deselect the transformation with TRAFOOF.
                 Clear alarm with RESET key. Restart part program.

17640   **Channel %1 block %2 spindle cannot be used as transformed axis %3**
Explanation: %1 = Channel number
                 %2 = Block number, label
                 %3 = Axis name, spindle number
                 The axis programmed for the spindle operation is involved in the current transformation as geometry axis. This is not allowed.
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: First switch off the transformation function.
                 Clear alarm with RESET key. Restart part program.

17800   **Channel %1 block %2 illegal fixed-stop end point programmed**
Explanation: %1 = Channel number
                 %2 = Block number, label
                 The position number n specified with the keyword FP=n is not permissible. Two absolute axis positions can be defined as fixed points via the axis-specific MD 30 600 **FIX_POINT_POS [n]**.
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Program keyword FP with machine fixed points 1 or 2.
                 Example:
                 Approach fixed point 2 with machine axes X1 and Z2.
                 N100 G75 FP=2 X1=0 Z2=0
                 Clear alarm with RESET key. Restart part program.

17900   **Channel %1 block %2 axis %3 use machine axis identifier**
Explanation: %1 = Channel number
                 %2 = Block number, label
                 %3 = Axis name, spindle number
                 At this point, the block context calls for a machine axis.
                 This is the case with:
                 • G74 (reference point approach)
                 • G75 (fixed point approach). If a geometry or additional axis identifier is used, then it must also be allowed as machine axis identifier.
                 (MD: 10000 AXCONF_MACHAX_NAME_TAB).
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Use machine axis identifier when programming.
                 Clear alarm with RESET key. Restart part program.

18001   **Channel %1 block %2 wrong definition of global protection area %3, error code %4**
Explanation: %1 = Channel number
                 %2 = Block number, label
                 %3 = Number of global protection zone
                 %4 = Error specification
                 There is an error in the definition of the protection area. The error numbers indicate the specific reason for the alarm. The following meanings apply:
                 1: Incomplete or conflicting contour definition
                 2: Contour encompasses more than one surface area
                 3: Tool-related protection zone is not convex.
                 4: If both boundaries are active in the 3rd dimension of the protection zone and both limits have the same value.
                 5: The number of the protection area does not exist (negative number, zero or greater than the maximum number of protection zones)
                 6: Protection zone definition consists of more than 10 contour elements
                 7: Tool-related protection zone is defined as inside protection zone.
                 8: Incorrect parameter used.
                 9: Protection zone to be activated is not defined
                 10: Incorrect modal G code used for protection zone definition.
                 11: Contour definition incorrect or frame activated.
                 12: Other errors not specified further
Reaction: Alarm display. Interface signals are set. NC Start disable.
Remedy: Modify definition of the protection zone and check MD. Clear alarm with Cancel key. No further operator action

emco

necessary.

18003   **Channel %1 block %2 channel-specific protection area %3 cannot be activated, error code %4**
Explanation: %1 = Channel number
                  %2 = Block number, label
                  %3 = Number of the channel-specific protection zone
                  %4 = Error specification
                  An error has occurred on activating the protection zone.The error number gives the specific reason for the alarm.
                  The following meanings apply:
                  1: Incomplete or conflicting contour definition.
                  2: Contour encompasses more than one surface area.
                  3: Tool-related protection zone is not convex.
                  4: If both boundaries are active in the 3rd dimension of the protection zone and both limits have the same value.
                  5: The number of the protection area does not exist (negative number, zero or greater than the maximum number
                  of protection zones).
                  6: Protection zone definition consists of more than 10 contour elements.
                  7: Tool-related protection zone is defined as inside protection zone.
                  8: Incorrect parameter used.
                  9: Protection zone to be activated is not defined.
                  10: Error in internal structure of the protection zones.
                  11: Other errors not specified further.
                  12: The number of protection zones simultaneously active exceeds the maximum number (channel-specific
                  machine data).
                  13,14: Contour element for protection zones cannot be created.
                  15,16: No more memory space for the protection zones.
                  17: No more memory space for the contour elements.
Reaction: Alarm display. Interface signals are set. Correction block is reorganized. Interpreter stop. NC Start disable.
Remedy: 1. Reduce the number of simultaneously active protection zones (MD).
                  2. Modify part program.
                  • Delete other protection zones.
                  • Preprocessing stop.
                  Clear alarm with NC Start and continue program.

18006   **Channel %1 block %2 serious error in definition of channel-specific protection area %3.**
Explanation: %1 = Channel number
                  %2 = Block number, label
                  %3 = Protection zone number
                  The protection zone definition must be terminated with EXECUTE before a preprocessing stop is performed. This
                  also applies to any that are initiated implicitly such as with G74, M30, M17.
Reaction: Alarm display. Interface signals are set. Correction block is reorganized. NC Start disable.
Remedy: Modify part program.
                  Clear alarm with NC Start and continue program.

18100   **Channel %1 block %2 invalid argument passed to FXS**
Explanation: %1 = Channel number
                  %2 = Block number, label
                  The following values are valid at the present time:
                  0: „Deselect traverse against fixed stop"
                  1: „Select traverse against fixed stop".
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Clear alarm with RESET key. Restart part program.

18101   **Channel %1 block %2 invalid argument passed to FXST**
Explanation: %1 = Channel number
                  %2 = Block number, label
                  Only the range 0.0 - 100.0 is valid at the present time.
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Clear alarm with RESET key. Restart part program.

18102   **Channel %1 block %2 invalid argument passed to FXSW**
Explanation: %1 = Channel number
                  %2 = Block number, label
                  Only positive values including zero are valid at the present time.
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable.
Remedy: Clear alarm with RESET key. Restart part program.

18300   **Channel %1 block %2 frame: Fine shift not possible**
Explanation: %1 = Channel number
                  %2 = Block number, label
                  Allocation of a fine shift to settable frames or the basic frame is not possible since MD $MN_FRAME_FINE_TRANS
                  is unequal to 1.
Reaction: Alarm display. Interface signals are set. Interpreter stop. NC Start disable
Remedy: Modify program or set MD $MN_FRAME_FINE_TRANS to 1.
                  Clear alarm with NC Start and continue program.

20000   **Channel %1 axis %2 reference cam not reached**

Explanation:	%1 = Channel number
	%2 = Axis name, spindle number
	After starting the reference point approach, the rising edge of the reduction cam must be reached within the section defined in the MD 34030 **REFP_MAX_CAM_DIST** (phase 1 of referencing). (This error occurs **only with incremental encoders**).
Reaction:	Alarm display. Interface signals are set. NC Stop when alarm. NC Start disable.
Remedy:	There are 3 possible causes of error:
	1. The value entered in MD 34030 **REFP_MAX_CAM_DIST** is too small. Determine the maximum possible distance from the beginning of reference motion up to the reduction cam and compare with the value in the MD: **REFP_MAX_CAM_DIST**, increase the value in the MD if necessary.
	2. The cam signal is not received by the PLC input module. Operate the reference point switch by hand and check the input signal on the NC/PLC interface (route: switch!connector!cable! PLC input!user program).
	3. The reference point switch is not operated by the cam. Check the vertical distance between reduction cam and activating switch.
	Clear alarm with RESET key. Restart part program.

20001   **Channel %1 axis %2 cam signal missing**

Explanation:	%1 = Channel number
	%2 = Axis name, spindle number
	At the beginning of phase 2 of reference point approach, the signal from the reduction cam is no longer available. Phase 2 of reference point approach begins when the axis remains stationary after deceleration to the reduction cam. The axis then starts in the opposite direction in order to select the next zero marker of the measuring system on leaving the reduction cam or approaching it again (negative/positive edge).
Reaction:	Alarm display. Interface signals are set. NC Stop when alarm. NC Start disable.
Remedy:	Check whether the **deceleration path** after the approach velocity is greater than the distance to reference point cam - in which case the axis cannot stop until it is beyond the cam. Use longer cam or reduce the approach velocity in machine data 34020 **REFP_VELO_SEARCH_CAM**. When the axis has stopped at the cam, it must be checked whether the signal „DECELERATION REFERENCE POINT APPROACH" is still available at the interface to the NCK (DB 31 - 48, DBX 12.7).
	• Hardware: Wire break? Short circuit?
	• Software: User program?
	Clear alarm with RESET key. Restart part program.

20002   **Channel %1 axis %2 zero reference mark not found**

Explanation:	%1 = Channel number
	%2 = Axis name, spindle number
	The zero marker of the **incremental encoder** is not within a defined section. Phase 2 of reference point approach ends when the zero marker of the encoder has been detected after the rising/falling edge of the PLC interface signal „DECELERATION REFERENCE POINT APPROACH" (DB 31 - 48, DBX 12.7) has given the trigger start. The maximum distance between the trigger start and the zero marker that follows is defined in the machine data 34060 **REFP_MAX_MARKER_DIST**. The monitor prevents a zero marker signal from being overtraveled and the next being evaluated as reference point signal. (Faulty cam adjustment or excessive delay by the PLC user program.)
Reaction:	Alarm display. Interface signals are set. NC Stop when alarm. NC Start disable.
Remedy:	Check the cam adjustment and make sure that the distance is sufficient between the end of the cam and the zero marker signal that follows. The path must be greater than the axis can cover in the PLC cycle time. Increase the machine data 34060 **REFP_MAX_MARKER_DIST**, but do not select a value greater than the distance between the 2 zero markers. This might result in the monitor being switched off.
	Clear alarm with the RESET key. Restart part program.

20003   **Channel %1 axis %2 encoder error**

Explanation:	%1 = Channel number
	%2 = Axis name, spindle number
	In a measuring system with distance-coded reference marks, the distance between two adjacent markers has been found to be more than twice the distance entered in the machine data 34300 **ENC_REFP_MARKER_DIST**. The control issues the alarm after having made a second attempt in reverse direction with half the traversing velocity and detecting that the distance is too large again.
Reaction:	Alarm display. Interface signals are set. NC Stop when alarm. NC Start disable.
Remedy:	Determine the distance between 2 odd reference point markers (reference point marker interval). This value (which is 20.00 mm on Heidenhain scales) must be entered in the machine data 34300 **ENC_REFP_MARKER_DIST**. Check the **reference point track** of the scale including the electronics for the evaluation.
	Clear alarm with RESET key. Restart part program.

20004   **Channel %1 axis %2 reference mark missing**

Explanation:	%1 = Channel number
	%2 = Axis name, spindle number
	In the **distance-coded length measurement system** two reference marks were not found within the defined searching distance (axis-specific MD: 34060 **REFP_MAX_MARKER_DIST**). No reduction cam is required for distance-coded scales (but an existing cam will be evaluated). The conventional direction key determines the direction of search. The searching distance 34060 **REFP_MAX_MARKER_DIST,** within which the two reference point markers are expected is counted commencing at the start point.
Reaction:	Alarm display. Interface signals are set. NC Stop when alarm. NC Start disable.
Remedy:	Determine the **distance** between 2 odd reference point markers (reference point marker interval). This value (which

is 20.00 mm on Heidenhain scales) must be entered in the machine data 34060 **REFP_MAX_MARKER_DIST**.
Check the **reference point track** of the scale including the electronics for the evaluation.
Clear alarm with RESET key. Restart part program.

20005	**Channel %1 axis %2 reference point approach aborted**
Explanation:	%1 = Channel number
	%2 = Axis name, spindle number
	Channel-specific referencing could not be completed for all specified axes (e.g. termination because of missing encoder enable, measuring system switchover, release of direction key, etc.).
Reaction:	Alarm display. Interface signals are set. NC Stop when alarm. NC Start disable.
Remedy:	Check the possible reasons for termination:

• Servo enable missing (DB 21 - 28, DBX 2.1)
• Measuring system switchover (DB 21 - 28, DBX 1.5 and DBX 1.6)
• Traversing key + or - missing (DB 21 - 28, DBX 8.6 and DBX 8.7)
• Feed override = 0
The axis-specific MD 34110 **REFP_CYCLE_NR** determines which axes are involved in the channel-specific referencing.
-1: No channel-specific referencing, NC Start **without** referencing.
0: No channel-specific referencing, NC Start **with** referencing.
1-8: Channel-specific referencing. The number entered here corresponds to the referencing sequence. (When all axes with contents 1 have reached the reference point, then the axes with contents 2 start, etc.).
Clear alarm with RESET key. Restart part program.

20006	**Channel %1 axis %2 reference point creep velocity not reached**
Explanation:	%1 = Channel number
	%2 = Axis name, spindle number
	In phase 2 of reference point approach (wait for zero mark), the cam end was reached but the reference point approach velocity was not within the tolerance window. (This can occur when the axis is already at the end of the cam at the beginning of reference point approach. This means that phase 1 has already been concluded and will not be started.) Phase 2 is terminated (this time in front of the cam) and reference point approach is started automatically once again with phase 1. If the approach velocity is not reached during the 2nd attempt, then referencing is aborted and the alarm is output.
	Approach velocity: 34040 **REFP_VELO_SEARCH_MARKER**
	Velocity tolerance: 35150 **SPIND_DES_VELO_TOL**
Reaction:	Alarm display. Interface signals are set. NC Stop when alarm. NC Start disable.
Remedy:	Reduce the MD for the approach velocity 34040 **REFP_VELO_SEARCH_MARKER** and/or increase the MD for the velocity tolerance 35150 **SPIND_DES_VELO_TOL**.
	Clear alarm with RESET key. Restart part program.

20007	**Channel %1 axis %2 reference point approach needs 2 encoders**
Explanation:	%1 = Channel number
	%2 = Axis name, spindle number
	Bei der Einstellung 34200 ENC_REFP_MODE = 6 werden 2 Geber benötigt!
Reaction:	Alarm display. Interface signals are set. NC Stop when alarm. NC Start disable.
Remedy:	Referiermodus 34200 ENC_REFP_MODE ändern o. zweiten Geber einbauen und konfigurieren
	Clear alarm with RESET key. Restart part program.

20008	**Channel %1 axis %2 Referenzpunktfahren benoetigt zweites referiertes Messsystem**
Explanation:	%1 = Channel number
	%2 = Axis name, spindle number
	2 encoders are needed for setting 34200 ENC_REFP_MODE = 6!.
Reaction:	Alarm display. Interface signals are set. NC Stop when alarm. NC Start disable.
Remedy:	Modify reference mode 34200 ENC_REFP_MODE or install and configure a second encoder
	Clear alarm with RESET key. Restart part program.

20050	**Channel %1 axis %2 handwheel mode active**
Explanation:	%1 = Channel number
	%2 = Axis name, spindle number
	The axes cannot be traversed in JOG mode using the traversing keys because traversing is still taking place via the The axes cannot be traversed in JOG mode using the traversing keys because traversing is still taking place via the handwheelhandwheel.
Reaction:	Alarm display.
Remedy:	Decide whether the axis is to be traversed by means of the jog keys or via the handwheel. End handwheel travel and delete the axial distance-to-go if necessary (interface signal DB 31 - 48, DBX 2.2).
	Alarm display showing cause of alarm disappears. No further operator action.

20051	**Channel %1 axis %2 handwheel mode not possible**
Explanation:	%1 = Channel number
	%2 = Axis name, spindle number
	The axis is already traveling via the traversing keys, so handwheel mode is no longer possible.
Reaction:	Alarm display.
Remedy:	Decide whether the axis is to be traversed by means of the jog keys or via the handwheel.
	Alarm display showing cause of alarm disappears. No further operator action.

20052	**Channel %1 axis %2 already active**
Explanation:	%1 = Channel number
	%2 = Axis name, spindle number

The axis is to traverse as machine axis in JOG mode via the jog keys on the machine control panel. However, this is not possible because:
1. It is already traversing as geometry axis (through the channel-specific interface DB 21 - 28, DBX 12.6, DBX 12.7, DBX 16.6, DBX 16.7 or DBX 20.6 and DBX 20.7) or
2. It is already traversing as machine axis (through the axis-specific interface DB 31 - 48, DBX 8.6 and DBX 8.7) or
3. A frame is valid for a rotated coordinate system and anoth er geometry axis involved in this is already traversing in JOG mode by means of the direction keys.

Reaction:	Alarm display.
Remedy:	Stop traversing through the channel or axis interface or stop the other geometry axis.
	Clear alarm with Cancel key. No further operator action necessary.

20053   **Channel %1 axis %2 DRF, FTOCON, external setting of offset not possible**
Explanation: %1 = Channel number
                  %2 = Axis name, spindle number
                  The axis is traversed in a mode (e.g. referencing) that allows no additional overlaid interpolation.
Reaction: Alarm display.
Remedy: Wait until the axis has reached its reference position or terminate reference point approach with „Reset" and start DRF once again.
                  Clear alarm with Cancel key. No further operator action necessary.

20054   **Channel %1 axis %2 wrong index for indexing axis in JOG mode**
Explanation: %1 = Channel number
                  %2 = Axis name, spindle number
                  1. The displayed indexing axis is to be traversed incrementally in JOG mode (by 1 indexing position). However, no further indexing position is available in the selected direction.
                  2. The axis is stationary at the last indexing position. In incremental traversing the working area limitation or the software limit switch is reached without an indexing position being located in front of it at which a stop could be made.
Reaction: Alarm display.
Remedy: Correct (add to) the list of indexing positions by means of the machine data
                  MD 10 900: **INDEX_AX_LENGTH_POS_TAB_1**
                  MD 10 910: **INDEX_AX_POS_TAB_1**
                  MD 10 920: **INDEX_AX_LENGTH_POS_TAB_2**
                  MD 10 930: **INDEX_AX_POS_TAB_2**
                  or set the working area limits or the software limit switches to other values.
                  Clear alarm with Cancel key. No further operator action necessary.

20055   **Channel %1 master spindle not available in JOG mode**
Explanation: %1 = Channel number
                  The displayed axis is to be traversed as machine axis in JOG mode with revolutional feed, but no master spindle has been defined from which the actual speed could have been derived.
Reaction: Alarm display. Interface signals are set.
Remedy: If the revolutional feed is also to be active in JOG mode, then a master spindle must be declared via the channel-specific machine data 20090 **SPIND_DEF_MASTER_SPIND**. In this case you have to open a screen in the **PARAMETER** operating area with the softkeys "SETTINGDATA" and "JOG DATA" and preselect the G function **G95** there. The JOG feedrate can then be entered in [mm/rev]. (If 0 mm/rev is set as JOG feed, the control takes the value assigned in the axis-specific MD 32050 **JOG_REV_VELO** or in the case of rapid traverse overlay 32040 **JOG_REV_VELO_RAPID).**
                  The revolutional feed in JOG mode is deactivated by changing the G function from **G95** to **G94**.
                  Clear alarm with Cancel key. No further operator action necessary.

20056   **Channel %1 axis %2 no revolutional feedrate possible. Axis/spindle %3 stationary**
Explanation: %1 = Channel number
                  %2 = Axis name, spindle number
                  %3 = Axis name, spindle number
                  An axis is to travel in JOG with revolutional feed, but the spindle/axis the feed is to be derived from is 0.
Reaction: Alarm display
Remedy: Traverse the spindle/axis from which the feed is to be derived.
                  Alarm display showing cause of alarm disappears. No further operator action required.

20057   **Channel %1 block %2 revolution velocity of axis/spindle %3 is less or equal zero.**
Explanation: %1 = Channel number
                  %2 = Block number, label
                  %3 = Axis name, spindle number
                  Revolutional feed has been programmed for an axis/spindle, but the velocity was not programmed or the programmed value is smaller than or equal to zero.
Reaction: Alarm display. Interface signals are set. NC Stop when alarm. NC Start disable. LOCALREACTION. COMPBLOCKWITHREORG. Channel processing not ready
Remedy: • Correct the part program or
                  • Specify the correct feed for PLC axes at the VDI interface, or
                  • Specify feed for oscillating axes in the setting data $SA_OSCILL_VELO.
                  Clear alarm with the RESET key. Restart part program.

20060   **Channel %1 axis %2 cannot move as geometry axis**
Explanation: %1 = Channel number

%2 = Achsname
The axis is currently not in „Geometry axis" state. Therefore, it cannot be traversed in JOG mode as geometry axis. If the abbreviation **WCS** (workpiece coordinate system) is displayed in the "Position" screen, then **only the geometry axes** can be traversed by means of the direction keys! (**MCS** ... Machine coordinate system; **all machine axes** can now be traversed by using the direction keys on the machine control panel).

Reaction: Alarm display.
Remedy: Check the operating steps to establish whether geometry axes really must be traversed, otherwise switch over to the machine axes by activating the "WCS/MCS" key on the machine control panel.
Clear alarm with Cancel key. No further operator action necessary.

20062   **Channel %1 axis %2 already active**
Explanation: %1 = Channel number
%2 = Axis name, spindle number
The displayed axis is already traversing as machine axis. Therefore, it cannot be operated as a geometry axis. Traversing of an axis can take place in JOG mode through 2 different interfaces.
1. As geometry axis: Through the channel-specific interface DB 21 - DB 28, DBX12.6 or DBX12.7
2. As machine axis: Through the axis-specific interface DB 31 - DB 48 DBX8.6 or DBX8.7
With the standard machine control panel, it is not possible to operate an axis as machine axis and geometry axis at the same time!
Reaction: Alarm display.
Remedy: Do not start the geometry axis until the traversing motion as machine axis has been concluded.
Clear alarm with Cancel key. No further operator action necessary.

20065   **Channel %1 master spindle not defined for geometry axes in JOG mode**
Explanation: %1 = Channel number
The displayed axis is to be traversed as geometry axis in JOG mode with rotary feed, but no master spindle has been defined from which the actual speed could be derived.
Reaction: Alarm display. Interface signals are set.
Remedy: If the revolutional feed is also to be active in JOG mode, then a master spindle must be declared via the channel-specific machine data 20090 SPIND_DEF_MASTER_SPIND. In this case you have to open a screen in the PARAMETER operating area with the softkeys "SETTINGDATA" and "JOG DATA" and preselect the G function **G95** there. The JOG feedrate can then be entered in [mm/rev]. (If 0 mm/rev is set as JOG feed, the control takes the value assigned in the axis-specific MD 32050 **JOG_REV_VELO** or in the case of rapid traverse overlay 32040 **JOG_REV_VELO_RAPID).**
The revolutional feed in JOG mode is deactivated by changing the G function from **G95** to **G94**.
Clear alarm with Cancel key. No further operator action necessary.

20070   **Channel %1 axis %2 programmed end position is beyond software limit %3**
Explanation: %1 = Channel number
%2 = Axis number
%3 = „+" or „-"
The axis is traversed as competing positioning axis and the target position is situated behind the corresponding software limit switch. The axis does not traverse.
Reaction: Alarm display.
Remedy: Specify smaller target position. Modify MD for SW limit switch. Possibly activate another SW limit switch.
Alarm display showing cause of alarm disappears. No further operator action necessary.

20071   **Channel %1 axis %2 programmed end position is beyond working area limit %3**
Explanation: %1 = Channel number
%2 = Axis number
%3 = „+" or „-"
The displayed axis is operated as a competing positioning axis. Its target position is behind the preset working area limitation.
Reaction: Alarm display.
Remedy: Parameterize target position within the permissible traversing range (parameter POS of FC ?) or correct position of software limit switch (activate 2nd software limit switch).
Alarm display showing cause of alarm disappears. No further operator action necessary.

20072   **Channel %1 axis %2 is not an indexing axis**
Explanation: %1 = Channel number
%2 = Axis number
The displayed axis is operated as a competing positioning axis. Its target position is parameterized in the FC INDEX-AXIS as indexing position number, but the axis is not an indexing axis.
Reaction: Alarm display.
Remedy: The FC POS-AXIS for linear and rotary axes should be used or the axis should be declared as an indexing axis.
Corresponding machine data for indexing axis declaration:
MD 30 500: **INDEX_AX_ASSIGN_POS_TAB**
MD 10 900: **INDEX_AX_LENGTH_POS_TAB_1**
MD 10 910: **INDEX_AX_POS_TAB_1**
MD 10 920: **INDEX_AX_LENGTH_POS_TAB_2**
MD 10 930: **INDEX_AX_POS_TAB_2**
Alarm display showing cause of alarm disappears. No further operator action necessary.

emco

20073   **Channel %1 axis %2 cannot be repositioned**
Explanation: %1 = Channel number
                 %2 = Axis number
                 The competing positioning axis cannot be positioned because it has already been restarted via the VDI interface and is still active. No repositioning motion takes place and the motion initiated by the VDI interface is not affected.
Reaction: Alarm display.
Remedy: None.
                 Clear alarm with Cancel key. No further operator action necessary.

20074   **Channel %1 axis %2 wrong index position**
Explanation: %1 = Channel number
                 %2 = Axis name, spindle number
                 For a competing positioning axis declared as indexing axis, the PLC has given an index number that is not available in the table.
Reaction: Alarm display.
Remedy: Check the indexing axis number given by the PLC and correct this if necessary. If the indexing axis number is correct and the alarm results from an indexing position table that has been set too short, check the machine data for indexing axis declaration.
                 MD 30 500: **INDEX_AX_ASSIGN_POS_TAB**
                 MD 10 900: **INDEX_AX_LENGTH_POS_TAB_1**
                 MD 10 910: **INDEX_AX_POS_TAB_1**
                 MD 10 920: **INDEX_AX_LENGTH_POS_TAB_2**
                 MD 10 930: **INDEX_AX_POS_TAB_2**
                 Alarm display showing cause of alarm disappears. No further operator action necessary.

20075   **Channel %1 axis %2 oscillating currently not possible**
Explanation: %1 = Channel number
                 %2 = Axis number
                 The axis cannot perform an oscillating movement now because it is already being traversed, e.g. in JOG mode.
Reaction: Alarm display.
Remedy: End the other traversing motion.
                 Clear alarm with Cancel key. No further operator action necessary.

20076   **Channel %1 axis %2 change of operation mode not possible during oscillation**
Explanation: %1 = Channel number
                 %2 = Axis number
                 The axis is performing an oscillating movement. Mode change is not possible because oscillation is not allowed in the selected mode.
Reaction: Alarm display. Interface signals are set. NC Stop when alarm. NC Start disable.
Remedy: Do not initiate mode change. Cause the PLC to check the axis and make sure in the PLC program that the axis ends oscillation if such mode changes take place.
                 Clear alarm with RESET key. Restart part program.

20077   **Channel %1 axis %2 programmed position is beyond software limit %3**
Explanation: %1 = Channel number
                 %2 = Axis number
                 %3 = „+" or „-"
                 The axis is traversed as oscillating axis and the target position (reversal position or end position) is located behind the corresponding software limit switch. The axis does not traverse.
Reaction: Alarm display. Interface signals are set. NC Start disable. NC Stop when alarm.
Remedy: Specify smaller target position. Modify MD for SW limit switch. Possibly activate another SW limit switch.
                 Clear alarm with RESET key. Restart part program.

20078   **Channel %1 axis %2 programmed position is beyond working area limit %3**
Explanation: %1 = Channel number
                 %2 = Axis number
                 %3 = „+" or „-"
                 The axis is traversed as oscillating axis and the target position (reversal position or end position) is located behind the corresponding valid working area limitation. The axis does not traverse.
Reaction: Alarm display. Interface signals are set. NC Start disable. NC Stop when alarm.
Remedy: Specify smaller target position. Deactivate working area limitation. Set working area limitation differentiall.
                 Clear alarm with RESET key. Restart part program.

20080   **Channel %1 axis %2 handwheel not assigned for overlaid handwheel motion**
Explanation: %1 = Channel number
                 %2 = Axis number
                 No handwheel has been assigned for this axis after handwheel overlay has been started in automatic mode.
Reaction: Alarm display.
Remedy: If handwheel control is required, a handwheel must be activated.
                 Alarm display showing cause of alarm disappears. No further operator action necessary.

20085   **Channel %1 contour handwheel: traverse direction or overtravel not allowed from beginning of block**
Explanation: %1 = Channel number
                 Travel takes place on the path with the contour handwheel in the opposite direction to the programmed travel direction and the starting point of the path has been reached at the start of the block.
Reaction: Alarm display

emco

Remedy:	Turn the contour handwheel in the opposite direction
	Alarm display verschwindet mit Alarmursache. No further operator action necessary.

20090 — **Axis %1 activation of fixed stop not possible. Check program line and axis parameters.**

Explanation: %1 = Axis name, spindle number
1. The „Traverse against fixed stop" function has been programmed with FXS[AX]=1 but the axis does not (yet) support this. Check MD 37000. This function is not available for gantry axes and simulated axes.
2. On selection, no movement was programmed for axis AX. AX is a machine axis identifier.
3. It is always necessary to program a traversing movement in the selection block for the axis/spindle for which the „Traverse against fixed stop" function is activated.
The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reaction: Mode group not ready.
In certain cases, it is possible to switch over for all channels via MD.
Channel not ready.
NC Start disable. NC Stop when alarm. Alarm display. Interface signals are set.

Remedy: • Check the axis type
• Check MD 37000
• Is a machine axis movement missing in the approach block?
Press the Reset key to clear alarm in all channels of this mode group.

20091 — **Axis %1 has not reached fixed stop**

Explanation: %1 = Axis name, spindle number
On attempting to traverse against a fixed stop, the programmed end position has been reached or the traversing movement has been aborted. The alarm can be concealed by means of the machine data $MA_FIXED_STOP_ALARM_MASK. The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reaction: Mode group not ready.
In certain cases, it is possible to switch over for all channels via MD.
Channel not ready.
NC Start disable. NC Stop when alarm. Alarm display. Interface signals are set.

Remedy: Correct the part program and the settings:
• Has the traversing block been aborted?
• If the axis position does not correspond to the programmed end position, then correct the end position.
• If the programmed end position is in the part, the triggering criterion must be checked.
• Has the contour deviation leading to triggering been dimensioned too large? Has the torque limit been set too high?
Press the Reset key to clear alarm in all channels of this mode group

20092 — **Axis %1 fixed stop mode still active**

Explanation: %1 = Axis name, spindle number
An attempt has been made to move an axis while it is in fixed stop or while the deselection function has not yet been completed.
The alarm can be reprogrammed in the MD ALARM_REACTION_CHAN_NOREADY (channel not ready).

Reaction: Mode group not ready.
In certain cases, it is possible to switch over for all channels via MD.
Channel not ready.
NC Start disable. NC Stop when alarm. Alarm display. Interface signals are set.

Remedy: Check the following:
• Has the axis at the fixed stop also been moved by a traversing movement of geometry axes?
• Is a selection carried out even though the axis is stationary at the stop?
• Has the deselection process been interrupted by a RESET?
• Has the PLC switched the acknowledgement signals?
Press the Reset key to clear alarm in all channels of this mode group.

20200 — **Channel %1 invalid spindle no. %2 with fine compensation of tool geometry**

Explanation: %1 = Channel number target channel
%2 = Spindle number
There is no spindle/axis assignment in the target channel for the spindle specified in the PUTFTOC command.

Reaction: Alarm display. Interpreter stop. Interface signals are set. NC Start disable. NC Stop when alarm.

Remedy: Modify program in channel that writes the tool fine compensation.
Clear alarm with RESET key. Restart part program.

20201 — **Channel %1 spindle %2 no tool assigned**

Explanation: %1 = Channel number
%2 = Spindle number
In order to make allowance for the fine tool compensation for the tool currently in the spindle, a spindle/tool assignment must be active. This is not presently the case for the programmed spindle in the target channel of fine tool compensation.

Reaction: Alarm display. Interpreter stop. Interface signals are set. NC Start disable. NC Stop when alarm.

Remedy: 1. Modify the part program (write the tool fine compensation).
2. Establish spindle/tool assignment by programming:
• TMON (tool monitoring).
• GWPSON (tool selection).
Clear alarm with RESET key. Restart part program.

emco

20203　　　　　　　　　　**Channel %1 no tool selected**
Explanation:　　　　　　　%1 = Channel number
　　　　　　　　　　　　　A tool fine compensation has been written for the active tool of channel %1 with PUTFTOC. No tool is active in this
　　　　　　　　　　　　　channel. Therefore, the compensation cannot be assigned.
Reaction:　　　　　　　　Alarm display. Interpreter stop. Interface signals are set. NC Start disable. NC Stop when alarm. Programm korrigieren
　　　　　　　　　　　　　Clear alarm with RESET key. Restart part program.

20204　　　　　　　　　　**Channel %1 instruction PUTFTOC not allowed during FTOCOF**
Explanation:　　　　　　　%1 = Channel number
　　　　　　　　　　　　　A tool fine compensation has been written for channel %1 with PUTFTOC. The tool fine compensation is not active
　　　　　　　　　　　　　in this channel. FTOCON must be active in the target channel of the PUTFTOC command.
Reaction:　　　　　　　　Alarm display. Interpreter stop. Interface signals are set. NC Start disable. NC Stop when alarm.
Remedy:　　　　　　　　Correct the program in the machining channel: Select FTOCON so that the channel is ready to receive the
　　　　　　　　　　　　　PUTFTOC command.
　　　　　　　　　　　　　Clear alarm with RESET key. Restart part program.

21617　　　　　　　　　　**Channel %1 block %2 transformation does not alow to traverse the pole.**
Explanation:　　　　　　　%1 = Channel number
　　　　　　　　　　　　　%2 = Block number, label
　　　　　　　　　　　　　The given curve passes through the pole or a forbidden area of transformation.
Reaction:　　　　　　　　Alarm display. Interface signals are set. NC Stop when alarm. NC Start disable.
Remedy:　　　　　　　　Modify part program (if alarm has occurred in AUTO mode). To escape from the alarm position, transformation must
　　　　　　　　　　　　　be deselected (it is not enough to try a RESET if the transformer remains active when RESET is applied).
　　　　　　　　　　　　　Clear alarm with RESET key. Restart part program.

21618　　　　　　　　　　**Channel %1 as from block %2 transformation active: overlaid motion too great**
Explanation:　　　　　　　%1 = Channel number
　　　　　　　　　　　　　%2 = Block number, label
　　　　　　　　　　　　　The share of overlaid motion on the transformation-related axes is so high that the path movement planned by the
　　　　　　　　　　　　　preparation no longer sufficiently corresponds to the actual ratio for the interpolation. Strategy of singularities,
　　　　　　　　　　　　　monitoring of working range limitation and dynamic Look Ahead are possibly no longer correct.
Reaction:　　　　　　　　Alarm display
Remedy:　　　　　　　　With overlaid motion it is necessary to keep a sufficiently large path safety distance with regard to poles and working
　　　　　　　　　　　　　range limitations.
　　　　　　　　　　　　　Clear alarm with Cancel key. No further operator action necessary.

21619　　　　　　　　　　**Channel %1 block %2 transformation active: motion not possible**
Explanation:　　　　　　　%1 = Channel number
　　　　　　　　　　　　　%2 = Block number, label
　　　　　　　　　　　　　The machine kinematics does not allow the specified motion.
Reaction:　　　　　　　　Alarm display. Interface signals are set. NC Stop when alarm. NC Start disable
Remedy:　　　　　　　　If the working area limitation is violated (see machine position), the part program's working area must be changed
　　　　　　　　　　　　　such that the possible operating range be adhered to (e.g. modified part settings). If the alarm is output in a pole
　　　　　　　　　　　　　position, care must be taken that in JOG it is only possible to traverse a pole or retract from it at the same angle at
　　　　　　　　　　　　　which it was entered.
　　　　　　　　　　　　　Note: RESET alone is not sufficient if Trafo also remains active after RESET.
　　　　　　　　　　　　　Clear alarm with RESET key. Restart part program.

21650　　　　　　　　　　**Channel %1 axis %2 overlaid motion not allowed**
Explanation:　　　　　　　%1 = Channel number
　　　　　　　　　　　　　%2 = Axis name, spindle number
　　　　　　　　　　　　　An overlaid motion was requested for the axis, however, this is not allowed due to the machine data
　　　　　　　　　　　　　FRAME_OR_CORRPOS_NOTALLOWED.
Reaction:　　　　　　　　Alarm display. Interface signals are set. NC Start disable. NC Stop when alarm.
Remedy:　　　　　　　　Deselect the overlaid motion or change machine data FRAME_OR_CORRPOS_NOTALLOWED
　　　　　　　　　　　　　Clear alarm with RESET key. Restart part program.

21700　　　　　　　　　　**Channel %1 block %3 axis %2 touch probe already deflected, edge**
Explanation:　　　　　　　%1 = Channel number
　　　　　　　　　　　　　%2 = Axis name, spindle number
　　　　　　　　　　　　　%3 = Block number
　　　　　　　　　　　　　The probe programmed under the keyword MEAS or MEAW is already deflected and has switched. For a further
　　　　　　　　　　　　　measuring operation, the probe signal must first be canceled (quiescent state of the probe). The axis display is of
　　　　　　　　　　　　　no significance at the present time but an axis-specific evaluation has been planned for later stages of development.
Reaction:　　　　　　　　Alarm display. Interface signals are set. NC Stop when alarm. NC Start disable.
Remedy:　　　　　　　　Verify the start position of the measuring operation or check the probe signals.
　　　　　　　　　　　　　Are the cables and connectors in good order?
　　　　　　　　　　　　　Clear alarm with RESET key. Restart part program.

21701　　　　　　　　　　**Channel %1 block %3 axis %2 measurement not possible**
Explanation:　　　　　　　%1 = Channel number
　　　　　　　　　　　　　%2 = Axis name, spindle number
　　　　　　　　　　　　　%3 = Block number
　　　　　　　　　　　　　Isn't measurement possible?
Reaction:　　　　　　　　Alarm display. Interface signals are set. NC Stop when alarm. NC Start disable.
　　　　　　　　　　　　　Clear alarm with RESET key. Restart part program.

EMCO

21702   **Channel %1 block %3 axis %2 measurement aborted**
Explanation: %1 = Channel number
                    %2 = Axis name, spindle number
                    %3 = Block number
                    The measurement block has ended (the programmed end position of the axis has been reached) but the activated touch probe has not yet responded.
Reaction: Alarm display.
Remedy: Verify the traversing movement in the measurements block.
                    • Is it necessary in all cases for the activated probe to have switched up to the specified axis position?
                    • Are the probe, cable, cable distributor, terminal connections in good order?
                    Clear alarm with Cancel key. No further operator action necessary.

21703   **Channel %1 block %3 axis %2 touch probe not deflected, edge polarity not possible**
Explanation: %1 = Channel number
                    %2 = Axis name, spindle number
                    %3 = Block number
                    The selected probe is not (!) deflected and therefore cannot record any measured value from the deflected to the non-deflected state.
Reaction: Alarm display. Interface signals are set. NC Stop when alarm. NC Start disable.
Remedy: - Check probe
                    - Check start positioning for measuring
                    - Check program
                    Clear alarm with RESET key. Restart part program.

22000   **Channel %1 block %3 spindle %2 change of gear stage not possible**
Explanation: %1 = Channel number
                    %2 = Spindelnummer
                    %3 = Block number, label
                    Automatic gear stage selection has been programmed with M40. The new M word is not in the present gear stage, but the spindle is not in „**Open-loop** control mode".
                    For automatic gear stage change (M40 in conjunction with spindle speed in address S) the spindle must be in "Open-loop control mode".
Reaction: Alarm display. Interface signals are set. NC Stop when alarm. NC Start disable.
Remedy: Before the S word which requires a gear stage change, change into the open-loop control mode of the spindle:
                    Change to the open-loop control mode is carried out with:
                    • M03, M04, M05 or M41 ... M45 from axis mode and positioning mode
                    • Interface signal "Gear is changed" (DB 31 - 48, DBX 16.3) from oscillation mode
                    Clear alarm with RESET key. Restart part program.

22010   **Channel %1 block %3 spindle %2 actual gear stage differs from requested gear stage.**
Explanation: %1 = Channel number
                    %2 = Spindelnummer
                    %3 = Block number, label
                    The requested gear stage change has been concluded. The actual gear stage reported by the PLC as being engaged is not the same as the required gear stage called for by the NC.
                    Note: Wherever possible, the requested gear stage should always be engaged..
Reaction: Alarm display.
Remedy: PLC-Programm korrigieren.
                    Clear alarm with Cancel key. No further operator action necessary.

22270   **Channel %1 block %2 spindle %3 spindle speed too high for thread cutting**
Explanation: %1 = Channel number
                    %2 = Block number, label
                    %3 = Axis name, spindle number
                    The spindle speed for thread cutting G33 is so high that the maximum axis velocity is exceeded because of the programmed thread lead.
Reaction: Alarm display.
Remedy: Program a lower spindle speed or a speed limitation with G26 S or reduce the spindle speed in front of the thread block by means of the setting data 43 220 SPIND_MAX_VELO_G26 or reduce the spindle override.
Remedy: Clear alarm with Cancel key. No further operator action necessary.

emco

Cycle Alarms 60000 - 63000

These alarms will be triggered by the machining
cycles of the control.
These are the same alarms as they would appear on
the original SIEMENS control.

61000　　　　　　　　　　　**No tool offset active**
Cycle:　　　　　　　　　　　LONGHOLE, SLOT1, SLOT2, POCKET1, POCKET2, CYCLE90, CYCLE93, CYCLE94, CYCLE95, CYCLE96.
Remedy:　　　　　　　　　　D offset must be programmed before the cycle is called.

61001　　　　　　　　　　　**Thread pitch wrong**
Cycle:　　　　　　　　　　　CYCLE84, CYCLE840, CYCLE96, CYCLE97.
Remedy:　　　　　　　　　　Check parameters for thread size and check pitch information (contradict each other).

61002　　　　　　　　　　　**Machining type incorrectly defined**
Cycle:　　　　　　　　　　　SLOT1, SLOT2, POCKET1, POCKET2, CYCLE93, CYCLE95, CYCLE97, CYCLE98.
Remedy:　　　　　　　　　　The value assigned to parameter VARI for the machining type is incorrect and must be altered.

61101　　　　　　　　　　　**Reference plane incorrectly defined**
Cycle:　　　　　　　　　　　CYCLE 81-90, CYCLE840, SLOT1, SLOT2, POCKET1, POCKET2, LONGHOLE.
Remedy:　　　　　　　　　　Either different values must be entered for the reference plane and the retraction plane if they are relative values
　　　　　　　　　　　　　　or an absolute value must be entered for the depth.

61102　　　　　　　　　　　**No spindle direction programmed**
Cycle:　　　　　　　　　　　CYCLE 86, CYCLE87, CYCLE88, CYCLE840
Remedy:　　　　　　　　　　Parameter SDIR (or SDR in CYCLE840) must be programmed.

61103　　　　　　　　　　　**Number of holes equals zero**
Cycle:　　　　　　　　　　　HOLES1, HOLES2
Remedy:　　　　　　　　　　No value has been programmed for the number of holes.

61104　　　　　　　　　　　**Contour violation of the slots/elongated holes**
Cycle:　　　　　　　　　　　SLOT1, SLOT2, LONGHOLE
Ursache:　　　　　　　　　　Incorrect parameterization of the milling pattern in the parameters that define the position of the slots/elongated
　　　　　　　　　　　　　　holes in the cycle and their shape.

61105　　　　　　　　　　　**Cutter radius too large**
Cycle:　　　　　　　　　　　SLOT1, SLOT2, POCKET1, POCKET2, LONGHOLE, CYCLE90
Remedy:　　　　　　　　　　The diameter of the milling cutter being used is too large for the figure that is to be machined; either a tool with a
　　　　　　　　　　　　　　smaller radius must be used or the contour must be changed.

61106　　　　　　　　　　　**Number of or distance between circular elements**
Cycle　　　　　　　　　　　HOLES2, LONGHOLE, SLOT1, SLOT2
Ursache:　　　　　　　　　　Incorrect parameterization of NUM or INDA, the circular elements cannot be arranged in a full circle.

61107　　　　　　　　　　　**First drilling depth incorrectly defined**
Cycle　　　　　　　　　　　CYCLE83
Ursache:　　　　　　　　　　First drilling depth is incompatible with final drilling depth.

61601　　　　　　　　　　　**Finished part diameter too small**
Cycle:　　　　　　　　　　　CYCLE94, CYCLE96
Ursache:　　　　　　　　　　A finished part diameter of <3 mm has been programmed

61602　　　　　　　　　　　**Tool width incorrectly defined**
Cycle:　　　　　　　　　　　CYCLE93
Ursache:　　　　　　　　　　Grooving tool is larger than the programmed groove width.

61603　　　　　　　　　　　**Groove form incorrectly defined**
Cycle　　　　　　　　　　　CYCLE93
Ursache:　　　　　　　　　　• Radii/chamfers at the base of the groove are not compatible with the groove width
　　　　　　　　　　　　　　• Face recess on a contour element that runs parallel to the longitudinal axis is not possible.

61604　　　　　　　　　　　**Active tool violates programmed contour**
Cycle:　　　　　　　　　　　CYCLE95
Remedy:　　　　　　　　　　Contour violation in relief cut elements as a result of the clearance angle of the tool being used, i.e. used a different
　　　　　　　　　　　　　　tool or check the contour subroutine.

EMCO

61605   **Contour incorrectly programmed**
Cycle: CYCLE95
Ursache: Illegal relief cut element detected.

61606   **Error on contour preparation**
Cycle: CYCLE95
Ursache: An error was detected during contour preparation, this alarm is always output with NCK alarm 10930 ... 10934, 15800 or 15810.

61607   **Starting point incorrectly programmed**
Cycle: CYCLE95
Ursache: The starting point reached before the cycle was called does not lie outside the rectangle described by the contour subroutine.

61608   **Wrong tool point direction programmed**
Cycle: CYCLE94, CYCLE96
Remedy: A tool point direction between 1 ... 4 that matches the undercut form must be programmed.

61109   **Form incorrectly programmed**
Cycle: CYCLE94, CYCLE96
Remedy: Check parameters for the undercut form

61110   **No drilling cycle active**
Cycle   HOLES1, HOLES2
Remedy: No drilling cycle was called modally before the drilling pattern cycle was called.

I 58