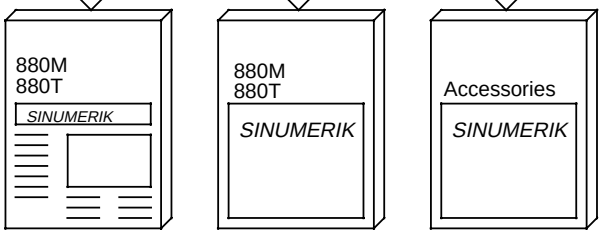
siemens

SINUMERIK 880, SW6
SINUMERIK 880 GA2, SW1

Operator's Guide 01.93 Edition

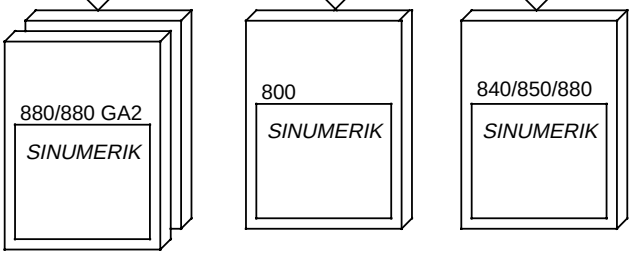
User Documentation

General Documentation



Technical data | Catalog NC 28 | Catalog NC 90

User Documentation

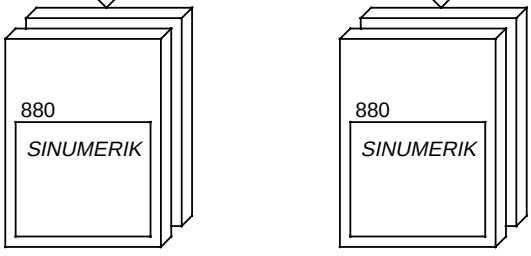


Operator's Guide
Programming Guide

Programming Guide
Cycles, UMS 4

Measuring Cycles
Version 20

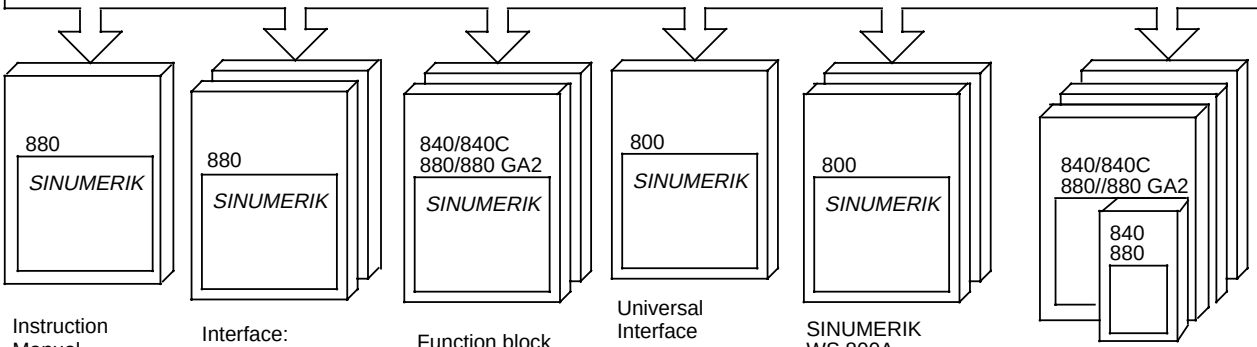
User/Manufacturer/Service Documentation



Function Manual:
- Electronic Gearbox
- Integrated Drive Control

Function Manual:
- Extended Functions for Spindles
- VT340 Terminal Emulator

Manufacturer Documentation



Instruction Manual

Interface:
- Signals
- Cables and Connections

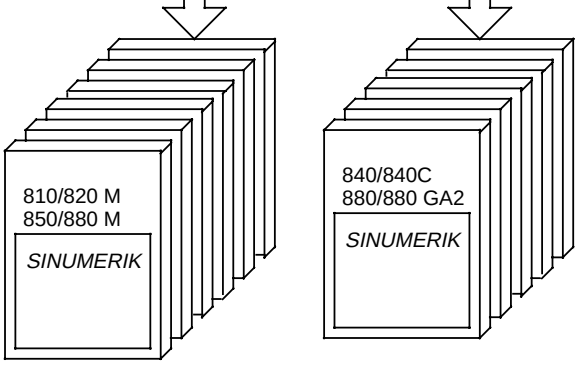
Function block Packages
Function Macros

Universal Interface

SINUMERIK WS 800A CL800 Cycle Language

PLC 135 WB
- Quick Reference
- Configuring
- S5-HLL
- PLC S5-155U Diff. Description

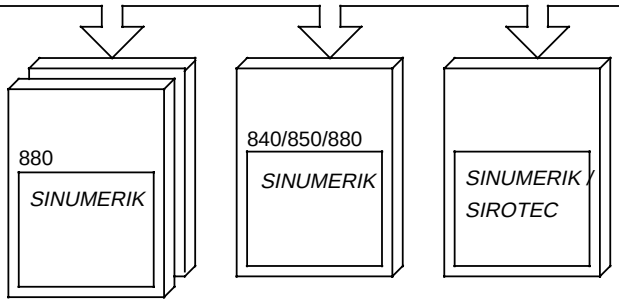
Manufacturer Documentation



Workshop UMS

Computer link

Service Documentation



Installation Guide
- Instructions
- Lists

Measuring Cycles
Version 20

Spare parts list

SINUMERIK 880, SW6
SINUMERIK 880 GA2, SW1

Operator's Guide

User Documentation

Valid for:

Control	*Software Version*
SINUMERIK 880 T/M	6
SINUMERIK 880 GA2 T/M	1

January 1993 Edition

SINUMERIK® documentation

Printing history

Brief details of this edition and previous editions are listed below.

The status of each edition is shown by the code in the ”Remarks” column.

Status code in ”Remarks” *column*:

A . . . New documentation
B . . . Unrevised reprint with new Order No.
C . . . Revised edition with new status.
If factual changes have been made on the page since the last edition, this is indicated by a new edition coding in the header on that page.

Edition	Order No.	Remarks	Software Version
05.91	6ZB5 410-0HC02-0BA0	**A**	6
01.93	6ZB5 410-0HC02-0BA1	**C**	6, GA2 SW1

Other functions not described in this documentation might be executable in the control. This does not, however, represent an obligation to supply such functions with a new control or when servicing.

This publication was produced on the Siemens 5800 Office System.
Subject to change without prior notice.

The reproduction, transmission or use of this document or its contents is not permitted without express written authority. Offenders will be liable for damages. All rights, including rights created by patent grant or registration of a utility model or design, are reserved.

© Siemens AG 1991 All Rights Reserved

Preliminary Remarks

Notes for the reader

The SINUMERIK documentation is organized in three parts:

- User Documentation
- Manufacturer Documentation and
- Service Documentation

This documentation has been written for machine tool **users**:

This publication applies to Software Version 6 and Basic Version 2, Software Version 1. Basic Version 2 is an advancement of Software Version 6. Basic Version 2 uses faster hardware with standard CPU modules for the COM and NC areas.

Depending on the software version, certain functions, general conditions, modules etc. will no longer be applicable. The following footnotes draw the reader's attention to such differences at the relevant points in the documentation:

c) up to and including Software Version 6
d) from GA2, Software Version 1

This publication provides detailed information required by the user for **operating** the SINUMERIK 880 control. It explains:

- The configuration of the control
- The operating elements:
  - CRT display with softkeys
  - Keyboard and display panel of the control
  - Keys and switches on the external machine control panel
- Operating sequences
- Data interfaces
- Diagnostics and maintenance

It is **not** possible to include in this Guide any details relating to an additional operator panel provided by the machine tool manufacturer.

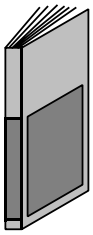
More information on other SINUMERIK 880 publications and on publications which are available for all SINUMERIK controls (”Universal Interface”, ”Measuring Cycles”, ...) are obtainable from your **Siemens local branch office**.

Different symbols are used in this publication to draw the reader's attention to important information:

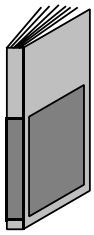
The symbol shown here appears when important technical facts or information are to be observed by the operator.

The symbol shown here appears in this documentation wherever the machine tool manufacturer can influence or modify the described function by changing a machine data (MD).

This symbol is intended to indicate that the function described can only be performed if the control contains the Option mentioned.



The symbol shown here appears in the documentation whenever a reference is made to other publications.



Technical comments

Other functions not described in this documentation might also be executable in the control. This does not, however, represent an obligation to supply such functions with a new control or when servicing.

This Operator's Guide applies to:
SINUMERIK 880, Software Version 6,
SINUMERIK 880 GA2, Software Version 1.

Configuration 1

Operation 2

Operating Sequences 3

Alarms and Notes for the Operator 4

Maintenance 5

Data Interfaces 6

Appendix 7

Contents

		Page
1	**Configuration**	1–1
1.1	Product	1–1
1.2	External machine control panel for SINUMERIK 880	1–2
1.3	Control without operator panel	1–3
2	**Operation**	2–1
2.1	Operating elements	2–1
2.1.1	SINUMERIK 880 operator interface	2–1
2.1.1.1	CRT display with softkeys	2–2
2.1.1.2	LED panel	2–4
2.1.1.3	Address keys	2–5
2.1.1.4	Symbol keys	2–5
2.1.1.5	Special characters	2–7
2.1.1.6	Arithmetic keys	2–7
2.1.1.7	Numerical keys	2–8
2.1.1.8	Editing and input keys	2–9
2.1.1.9	Control keys	2–10
2.1.1.10	Assignment of the full standard keyboard in NC mode	2–19
2.1.1.11	User-assignable keys	2–20
2.1.1.12	Operating mode group selection keys	2–20
2.1.2	Operator interface of the machine control panel for SINUMERIK 880	2–21
2.2	Switching the control on/off	2–33
2.2.1	Switching on the SINUMERIK 880	2–33
2.2.2	Switching off the SINUMERIK 880	2–33
2.3	Operating modes	2–34
2.3.1	General	2–34
2.3.2	Operating modes - overview	2–34
2.3.3	Selection of operating modes with external machine control panel	2–37
2.3.4	"RESET" with change of operating mode	2–38
2.3.4.1	No change of menu when switching between JOG/INC	2–39
2.3.4.2	Transformation states in the modes	2–40
2.3.5	Branching to functions within an operating mode	2–41
2.3.6	Jumping back to functions in higher-level menus within an operating mode	2–42
2.4	Operating mode menu trees	2–43
2.4.1	AUTOMATIC mode	2–44
2.4.2	JOG mode	2–45
2.4.3	JOG mode (after TEACH IN/PLAYBACK in AUTOMATIC mode)	2–45
2.4.4	MDI-AUTOMATIC mode	2–46
2.4.5	PRESET mode	2–46
2.4.6	INC FEED 1...10 000 mode	2–47
2.4.7	REFPOINT mode	2–47
2.4.8	REPOS mode	2–47
2.4.9	Operating functions	2–48
2.4.9.1	SETTING DATA	2–48
2.4.9.2	DATA IN-OUT	2–49

2.4.9.3 PART PROGRAM 2–50
2.4.9.4 DIAGNOSTICS 2–51
2.4.9.5 GUIDANCE 2–53
2.4.9.6 SIMULATION 2–57
2.5 Glossary of softkey functions 2–59

3 Operating Sequences 3–1
3.1 Preliminary remarks 3–1
3.2 Preparation 3–2
3.2.1 Switching on 3–2
3.2.2 Approach reference point (REFPOINT) 3–2
3.2.2.1 Setting-up mode, division-based 3–4
3.2.2.2 Semi-automatic drift compensation 3–6
3.2.3 Tools 3–7
3.2.4 Tool offset 3–7
3.2.4.1 Input of tool offset 3–7
3.2.4.2 Deleting/modifying an individual offset value 3–11
3.2.4.3 Deleting all offset values of a tool offset number D... 3–12
3.2.4.4 Recut thread 3–13
3.2.5 Zero offset 3–14
3.2.5.1 Settable zero offsets 3–17
3.2.5.2 Programmable zero offset 3–18
3.2.5.3 External zero offset 3–19
3.2.6 Electronic gear (ELG) 3–20
3.2.7 Synchro-spindle 3–33
3.2.8 Setting data 3–41
3.2.9 Program input 3–56
3.2.9.1 Program input with the keyboard 3–56
3.2.9.2 Program input with operator prompting 3–58
3.2.9.3 Plane selection 3–59
3.2.10 Contour 3–60
3.2.11 Program correction (EDIT):
Insert/modify/delete word, insert/delete block 3–63
3.2.11.1 Correction display ("CORRECTION BLOCK") 3–65
3.2.12 Program management 3–66
3.2.12.1 Displaying the stored programs (DIRECTORY) 3–66
3.2.12.2 Protection of subroutines (Cycle lock) 3–67
3.2.12.3 Cycle lock release (ENABLE) 3–68
3.2.12.4 Copying a program (COPY) 3–69
3.2.12.5 Moving cycles (MOVE) 3–70
3.2.12.6 Simultaneous editing of programs 3–71
3.2.12.7 Renaming a program (RENAME) 3–72
3.2.12.8 Deleting a program (DELETE) 3–73
3.2.12.9 Reorganizing the program memory 3–74
3.2.12.10 Selecting a program 3–75
3.2.13 Simulation 3–76
3.2.13.1 Simulation data 3–76
3.2.13.2 Input of simulation tool data 3–80
3.2.13.3 Simulation data (BLANK) 3–86
3.2.13.4 Simulation data (SETTING DATA) 3–87
3.2.13.5 Sequence of simulation 3–91
3.2.13.6 Selection of simulation window 3–92
3.2.13.7 Example of simulating a part program for SINUMERIK 880M 3–95
3.2.14 Data input/data output 3–96

3.2.14.1 Setting data bits 3–96
3.2.14.2 Data input 3–98
3.2.14.3 Data output 3–101
3.2.14.4 File transfer 3–104
3.3 Operation 3–105
3.3.1 Starting a part program 3–105
3.3.2 AUTOMATIC mode
Calling the ACTUAL VALUES or ACTUAL BLOCK display 3–106
3.3.3 Influencing AUTOMATIC operation 3–108
3.3.3.1 Influencing the program 3–109
3.3.3.2 Execution from external 3–115
3.3.3.3 OVERSTORE 3–117
3.3.3.4 BLOCK SEARCH 3–118
3.3.3.5 Interrupting the program 3–121
3.3.4 Manual data input-Automatic (MDI AUTOMATIC) mode 3–122
3.3.5 TEACH IN 3–123
3.3.6 PLAYBACK 3–124
3.3.7 JOG mode 3–127
3.3.8 INC FEED 1 ... INC FEED 10 000 mode (Incremental) 3–128
3.3.9 DRF 3–130
3.3.10 PRESET mode (Set actual value) 3–132
3.3.11 REPOS mode (Repositioning) 3–134
3.3.12 Scratch 3–137

4 Alarms and Notes for the Operator 4–1
4.1 Alarms 4–1
4.1.1 General 4–1
4.1.2 Display of alarms on the CRT 4–1
4.1.3 Display representation 4–2
4.1.4 Alarm numbers and groups/clearing alarms 4–5
4.1.5 DIAGNOSTICS/selection of further alarms 4–6
4.1.6 Effects of alarms 4–6
4.1.7 Listing of alarms/alarm description 4–7
4.1.8 Absolute encoder error 4–118
4.2 Notes for the operator 4–119
4.2.1 Listing of interactive dialog texts 4–119

5 Maintenance 5–1
5.1 Operating data 5–1
5.2 Replacing the battery 5–1
5.3 Memory submodule with back-up battery 5–2
5.4 Handling modules 5–3
5.5 Cleaning 5–4

6 Data Interfaces 6–1
6.1 General 6–1
6.2 Setting data for description of the interfaces 6–1
6.3 Assignment of the setting data for the connection of peripheral devices . 6–4
6.4 Device connection data 6–7

7 **Appendix** 7–1
7.1 List of abbreviations 7–1
7.2 List of terms used 7–3
7.3 SINUMERIK 880 Operator interface-
– Layout of control and machine control panel 7–12

1 Configuration

1.1 Product

The SINUMERIK 880 is a microprocessor-controlled NC continuous-path control system for complex machine tools.

((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-0HC01-0BA0
Seite 1–1 einmontieren))

SINUMERIK 880

- SINUMERIK 880T
  is used primarily to control turning machines.
- SINUMERIK 880M
  is used primarily to control boring**/**drilling and milling machines.
- Programming can be either computer-aided or manual.

© Siemens AG 1991 All Rights Reserved

- Operation:
  - Softkeys for selecting different softkey functions
  - 12" graphics screen[1)]
  - Symbol**/**numerical keyboard and function keys
- Screen displays provide information in plaintext, e.g. covering:
  - current NC operating modes
  - setpoint**/**actual values
  - NC and PLC alarms
    etc.
- Graphics displays aid the programmer when entering programs at the machine.
- "Blueprint programming" is available for higher-speed programming of complex contour elements.
- Entered programs may be simulated graphically (Option).
- The SINUMERIK 880
  has a part program memory of 32 K (32 000 punched tape characters) in the basic version.

1.2 External machine control panel for SINUMERIK 880

The SINUMERIK 880 basic version has no machine control panel!

A machine control panel is necessary to operate the control with the machine.
It is either supplied by the machine tool manufacturer as an **additional operator panel** or can be ordered from SIEMENS (Ordering Data - Option).

The basic version of the external machine control panel from SIEMENS is equipped with further important operating elements for the control to function with the machine:

- Emergency stop switch
- Operating mode selector switch
- Feedrate**/**rapid traverse override switch
- Spindle speed override switch
- Single block switchover
- Key-operated switch
- RESET key
- NC ON key (switching on control)
- ”Program start**/**Program stop” keys
- Feedrate start**/**feedrate stop keys
- Spindle start**/**spindle stop keys
- Axis selector switch
- Direction keys**/**rapid traverse key

In addition, the machine control panel can be supplemented with an expansion submodule. (2nd selector switch for additional auxiliary axes).

---

1) as from GA2, Software Version 1: color graphics screen (RMOS operator panel)

© Siemens AG 1991 All Rights Reserved

1.3 Control without operator panel

The "CONTROL WITHOUT OPERATOR PANEL" function is an option.

The "CONTROL WITHOUT OPERATOR PANEL" function enables you to operate the control and to switch it on and off without an operator panel. This function is chiefly used when several controls are available and the operator panel is only needed for control very rarely.

Sequence of operation

Disconnecting the operator panel

The operator panel is only allowed to be disconnected while NC operation is active. To disconnect the panel proceed as follows:

1. Switch on operation without operator panel using the relevant switch
2. Switch off the operator panel
3. Disconnect the operator panel (pull cable)

After the operator panel has been properly disconnected, the control can be arbitrarily switched on and off without the operator panel being used.

The switch for activating/deactivating the "CONTROL WITHOUT OPERATOR PANEL" function is machine-specific. Its position and designation are determined by the machine manufacturer.

Connecting the operator panel

To connect the operator panel while NC operation is taking place proceed in the following sequence:

1. Connect the operator panel (connect cable)
2. Switch on operator panel
3. Switch off operation without operator panel using the relevant switch

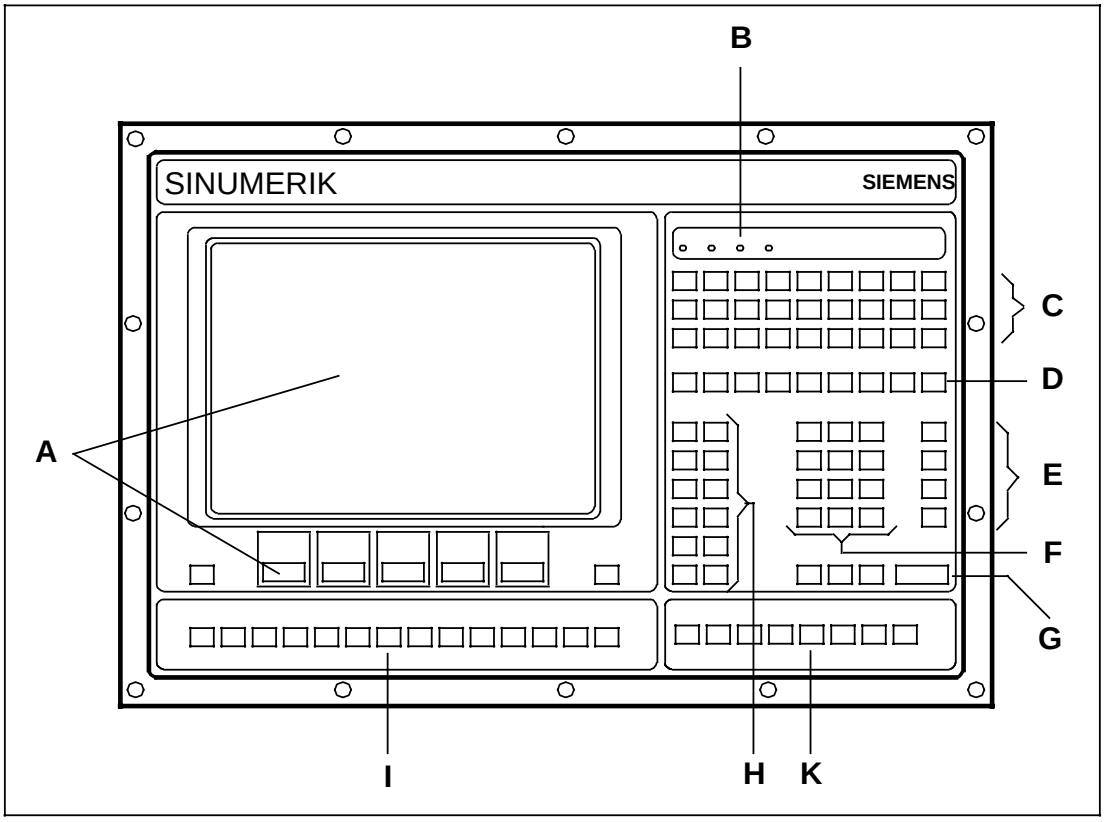
The control is always started up with the operator panel. The "CONTROL WITHOUT OPERATOR PANEL" function is deselected as long as control installation is being selected.

© Siemens AG 1991 All Rights Reserved

2 Operation

2.1 Operating elements

2.1.1 SINUMERIK 880 operator interface



SINUMERIK 880, Operator panel

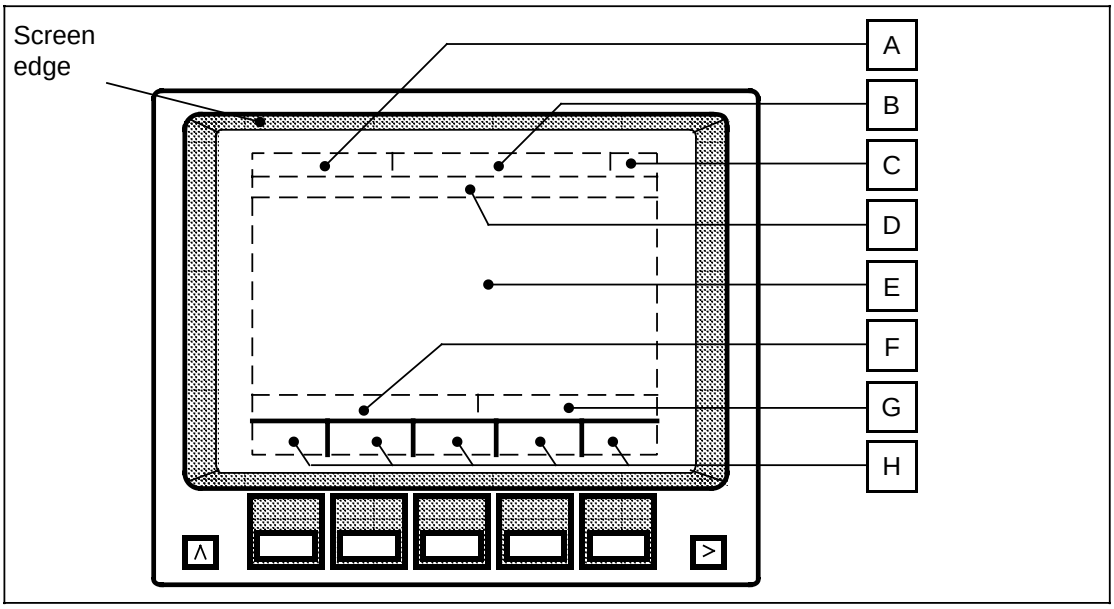
A 12" CRT display[1)] with 5 integrated softkeys (see Section 2.1.1.1)
B Display panel with 4 LEDs (see Section 2.1.1.2)
C Address keys (see Section 2.1.1.3)
D Symbol keys (see Section 2.1.1.4)
Special characters (see Section 2.1.1.5)
E Arithmetic keys (see Section 2.1.1.6)
F Numerical keys (see Section 2.1.1.7)
G Editing and input keys (see Section 2.1.1.8)
H Control keys (see Section 2.1.1.9)
I User-assignable keys (see Section 2.1.1.10)
K Keys for the selection of 8 operating mode groups (see Section 2.1.1.11)

---

1) as from GA2, Software Version 1: color graphics screen

© Siemens AG 1991 All Rights Reserved

2.1.1.1 CRT display with softkeys



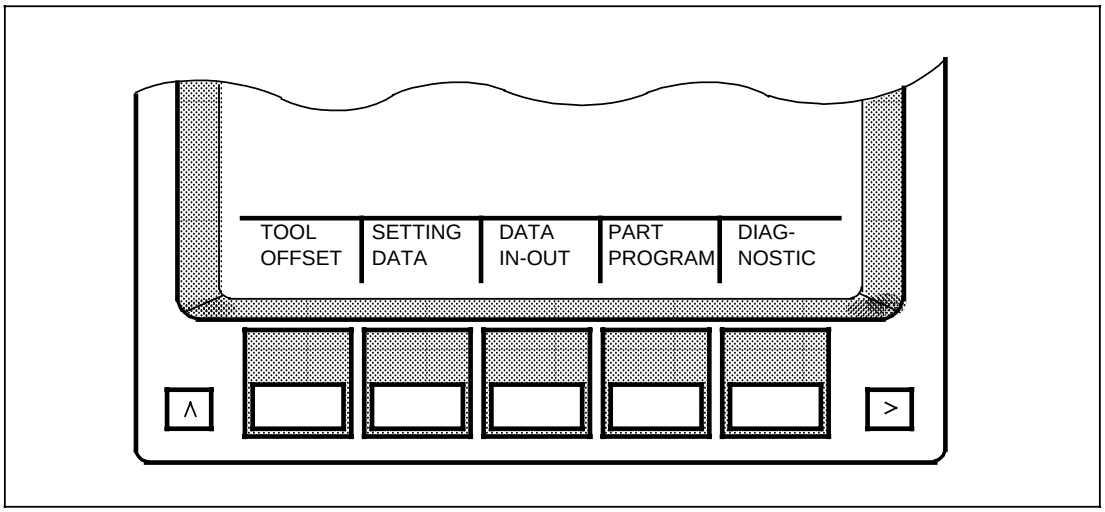
CRT display layout, text/graphics only within the area of the dotted lines

The CRT display is divided into **17 lines, each of 41 characters**. The following table shows the display function of the individual line/line area:

Note	CRT line ...	Display area for ...	Max. no. of characters
A	1	Operating mode selected	14
B		Operating state	23
C		Channel number and mode group number	4 [1])
D	2	Alarms and messages	41
E	3 to 14	NC displays: texts, graphics	12 x 41
F	15	Interactive dialog text/notes for the operator	24
G		Inputs from keyboard (numbers, symbols, etc.)	17
H	16 and 17	Softkey menu for up to 5 softkey functions	5 x 2 x 7

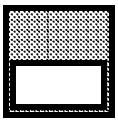
1) characters displayed in inverse video

© Siemens AG 1991 All Rights Reserved



Keys below the CRT display

Softkeys



With the operation of any one of the 5 softkeys (a **softkey** is defined as a key that does not have a fixed function), you select the required softkey function shown in the menu directly above the softkey.

RECALL Key for jump back to the text display of a higher-level menu



With the operation of this key, you change the softkey functions displayed by returning to a **higher-level menu**.

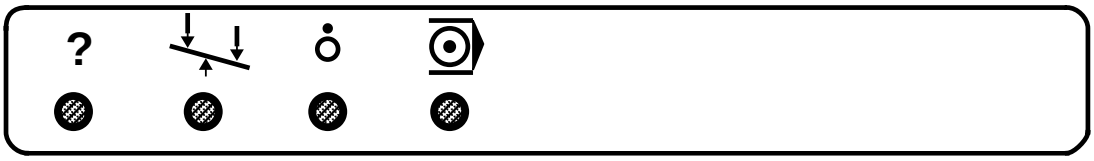
ETC Key for extension of the same menu



With the operation of this key, you change the softkey functions displayed. Additional functions in the **same menu** are displayed.

© Siemens AG 1991 All Rights Reserved

2.1.1.2 LED panel



View of LED panel

LED: ”Monitor responded/alarm”



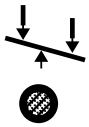
- The red LED lights up whenever there is a signal from the diagnostics monitor. The CRT display shows you the relevant alarm no., and the alarm message text (2nd line of display).
- The alarm numbers are explained in the Alarm list in Section 4 of this Guide.
- The LED is extinguished when the alarm has been acknowledged:

  with  "Acknowledge" key (Section 2.1.1.9)

  with  "RESET" key (Section 2.1.2)

  With certain messages, the display is only extinguished when the cause of the fault has been cleared (Section 4).

LED: ”Position not yet reached”



- The green LED is lit when at least one axis is moving.
- The LED is extinguished when all axes have reached their set positions.
- The LED remains lit even if none of the axes are moving, if the drift of one axis is too great.

LED: "Feed hold"



The red LED is lit when feed is interrupted.
The program is stopped.

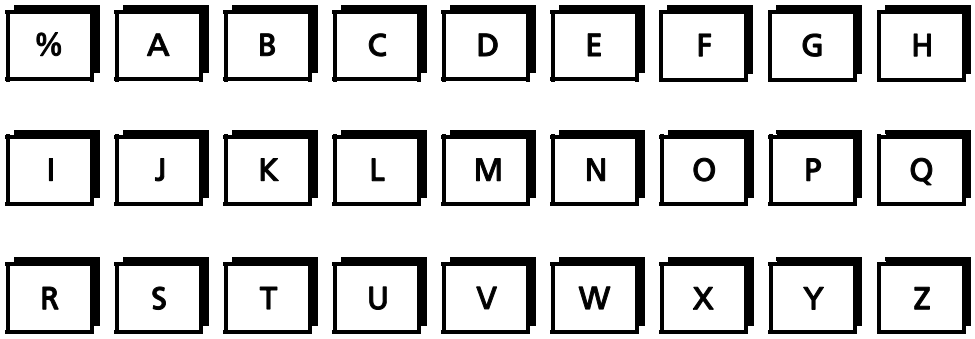
LED: "Program running"



- The green LED is lit when a program is being processed, even if the axes are not moving!
- The LED is extinguished when program processing is completed:
  – with programmed stop M00, M01
  – at the end of a single block
  – at the end of the program

© Siemens AG 1991 All Rights Reserved

2.1.1.3 Address keys



View of the address keys

Explanation of the key functions



Percent sign
Symbol for "Program start"



Letters A to Z

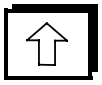
2.1.1.4 Symbol keys



View of the symbol keys

© Siemens AG 1991 All Rights Reserved

Explanation of the key functions



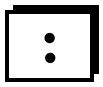
Shift key for lower-case letters
(a...f, for entering hexadecimal numbers)

To enter lower-case letters or special characters press the shift key until you have pressed the next key.

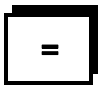


Skip block

When you use this key to put this symbol in front of blocks "N...", they are skipped/ignored when the program is processed if an external "Skip" switch is active!



Identifier for "Main block"



Equals sign
(for assignment of parameter values and axes with address expansion)



Address for "Program control function" (@-Code):

With the aid of the three-digit code "@ ..." you can, for example, **program cycles manually**.



Open parenthesis for start of coordination



Open parenthesis for start of comment



Pace character (blank)



End of block (LINE FEED)

© Siemens AG 1991 All Rights Reserved

2.1.1.5 Special characters

Explanation of key functions/key combinations

Required symbol	Key combination	Example of usage
, (comma)	⇧ :	---
#	⇧ =	@f22 Comment (#10)
	⇧ @	Input buffer e.g. R23= 00116
[	⇧ (	Parentheses for coordination command begin
]	⇧)	Parentheses for coordination command end

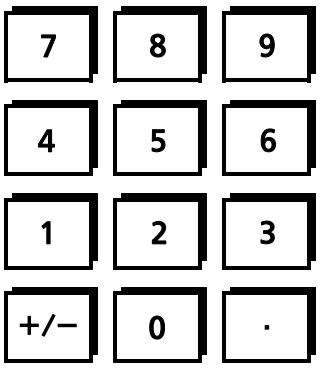
2.1.1.6 Arithmetic keys

View and explanation of key functions

÷	Division
*	Multiplication
–	Subtraction
+	Addition

© Siemens AG 1991 All Rights Reserved

2.1.1.7 Numerical keys



View of numerical keys

Explanation of the key functions

0 ... 9	Numerals 0 to 9.
+/−	Change sign This key reverses the sign: "+" becomes "-" and vice versa!
.	Decimal point

© Siemens AG 1991 All Rights Reserved

2.1.1.8 Editing and input keys



View of editing/input keys

Delete input/interactive dialog text (note for operator)



With this key, you delete:

- Characters on the **input line**
  - with **single** operation:
    always the last/most extreme right character
  - with **continuous operation**:
    all characters consecutively, from right to left, until the input line is clear.
- Interactive dialog text on the **operator message line**
  - with **single** operation: all characters simultaneously

Delete word/block



Delete word

- With this key you delete from the **part program memory** or a **list display** or an **input field**:
  The **word** on the CRT display to the **right** of the cursor (see Section 2.1.1.9), when **its address** is shown in the input line

Delete block

- With this key you delete from the **part program memory**:
  The **block** on the CRT display to the **right** of the cursor, when the **same block number** is shown on the input line.

Modify word

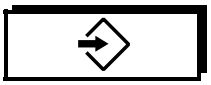


With this key you can modify a word in the **part program memory** or a **list display** or an **input field**:

- The **word** on the CRT display to the **right** of the cursor (see Section 2.1.1.9). The word marked with the cursor is modified to the word with the **same address** that is shown on the input line.
- For tool offsets the **value** can be modified **additively** with this key.

© Siemens AG 1991 All Rights Reserved

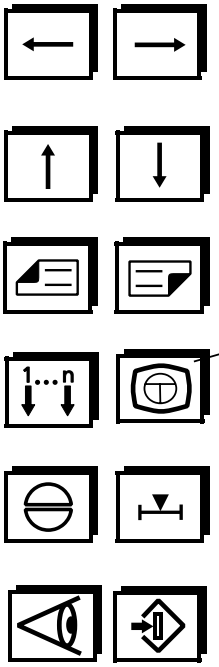
Input character/word



You conclude your inputs with this key:

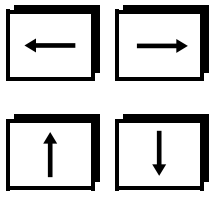
- The **characters** shown on the input line are transferred into the **list display** or the **input field** where the cursor (see Section 2.1.1.9) is positioned.
- A **word** shown on the input line is transferred into the **part program memory.**

2.1.1.9 Control keys



View of the control keys

Cursor left/right movement
Cursor up/down movement



With these keys you move the cursor on the CRT.
SINUMERIK 880 differentiates between:

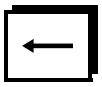
- Cursor for marking a word**/**block within displayed **part programs**

  Cursor appearance**/**format: ▮
- Cursor for marking an input field within **list display/input screen form**

Cursor appearance**/**format: ▬▬ or ▬
(Cursor length depends on the max. permitted number of characters of the marked field.)

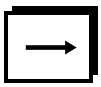
© Siemens AG 1991 All Rights Reserved

Cursor movement in a part program



With this key you move the cursor:

- from word to word to the left
- to the end of the previous block (after the last word/in front of L_F) when the cursor is positioned before the start of a block.



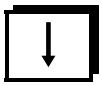
With this key you move the cursor:

- from word to word to the right
- before the start of the following block when the cursor is positioned at the end of a block (after the last word/in front of L_F).



With this key you move the cursor

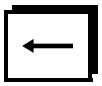
- from the start of block to the start of the previous block
- to the start of the same block if the cursor is positioned within a block
- to the start of program by continuously operating the key.



With this key you move the cursor

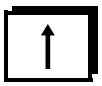
- from the start of block to the start of the following block
- to the start of the following block if the cursor is positioned within a block
- to the end of program by continuously operating the key.

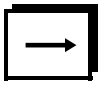
Cursor movement in an input screen form/dimensioned graphics display



With one of these two keys, you move the cursor

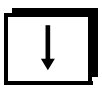
- **Backwards** in an **input screen form** [1]) from field to field:
  from right to left in the line and then to the line above from right to left etc.
- **Backwards** in a **dimensioned graphics display** in accordance with the specified data sequence;
  if the key is operated continuously the cursor jumps to the input field for the first dimension to be input.





With one of these two keys, you move the cursor

- **Forwards** in an **input screen form** [1]) from field to field:
  from left to right in the line, and then to the line below from left to right etc.
- **Forwards** in a **dimensioned graphics display**, in accordance with the specified data sequence;
  If the key is operated continuously, the cursor jumps to the input field for the last dimension to be input.

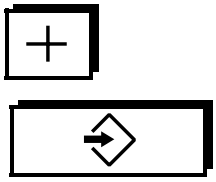


---

1) There are input screen forms for: tool offset, setting data, machine data, operator guidance

© Siemens AG 1991 All Rights Reserved

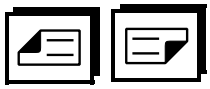
Switching off automatic cursor progression



If a value is entered in the input screen form and concluded with the "input" key, the cursor automatically jumps to the next input field.

To suppress the automatic movement of the cursor to the next field, press the + key and then the input key. The automatic cursor progression remains switched off until a cursor key is pressed again.

Paging up/down



By operating either of these keys you can change the current CRT display when further displays of similar format are available.



You page one display **up.**

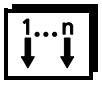


You page one display **down.**

In the case of SINUMERIK 880 GA2, Software Version 1: If the number of axes is increased to more than five, the sixth and further axes can be displayed in JOG, INC and AUTOMATIC modes by pressing the "page down" key.

© Siemens AG 1991 All Rights Reserved

Channel switchover



- By operating this key once, you switch to the next channel no. up from the one displayed on the right in the top line of the display.
- Operating the key again switches to the next channel up, etc.

Explanation:

The NC area of the SINUMERIK 880 is divided into several channels.
Each channel can be regarded as a standalone NC, with:

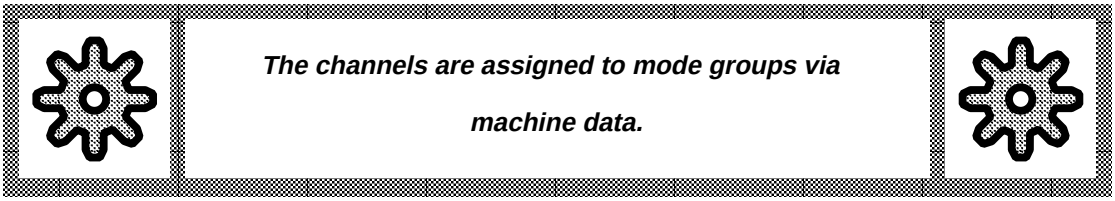
- its own program
- its own decoding
- its own interpolation

Each channel has its own block preparation which combines data from the part program with NC data (zero offsets, tool offsets etc.) and transfers them to the interpolator via a buffer memory.

- **SINUMERIK 880** : the NC area is divided into max. **16 channels.**
- **SINUMERIK 880 GA2**: the NC area is divided into max. **8 channels.**

The **basic version** has **4 channels.**

The NC channels are grouped together in **operating mode groups**.
With SINUMERIK 880 **eight mode groups** can be selected via keys (see Section 2.1.1.12).

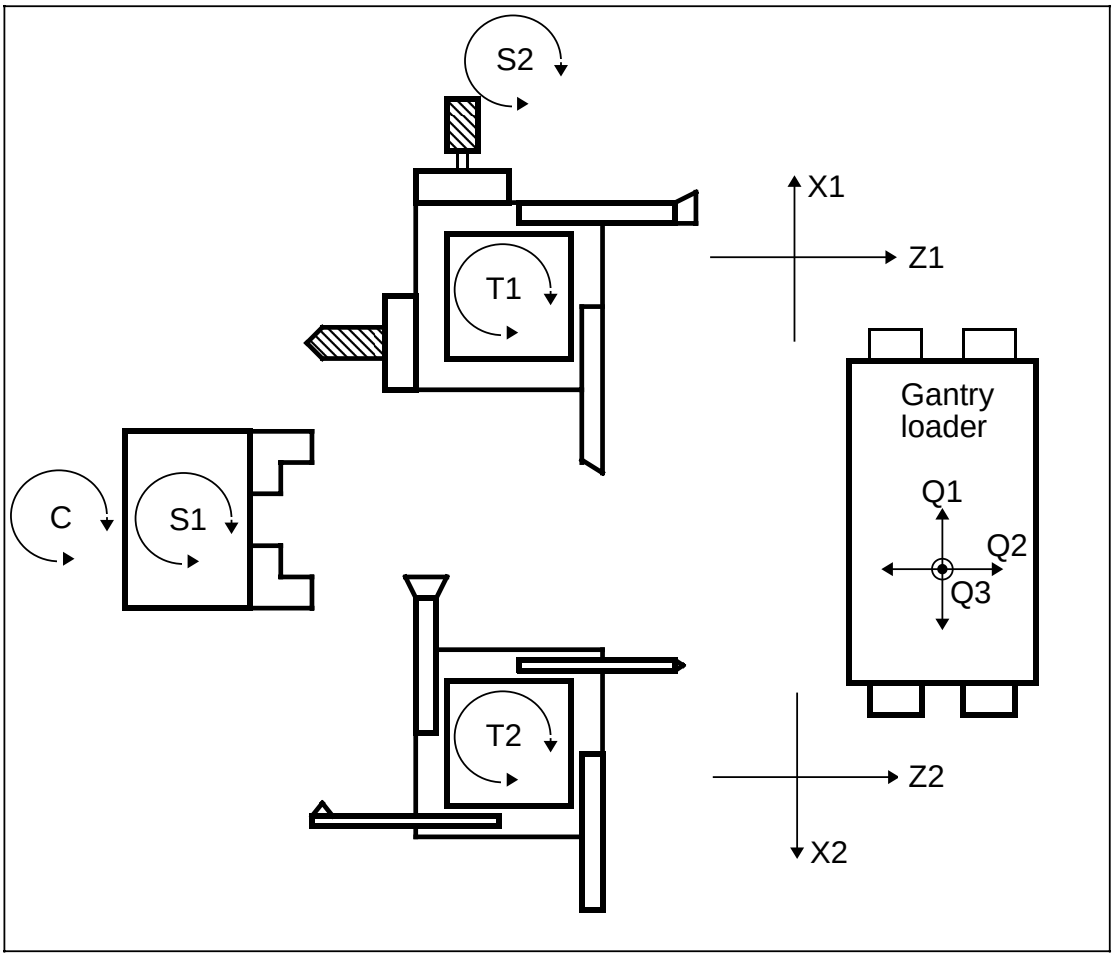


© Siemens AG 1991 All Rights Reserved

Depending on the workpiece to be machined, machining programs can be created for all channels and read into the central NC program memory.

Example:

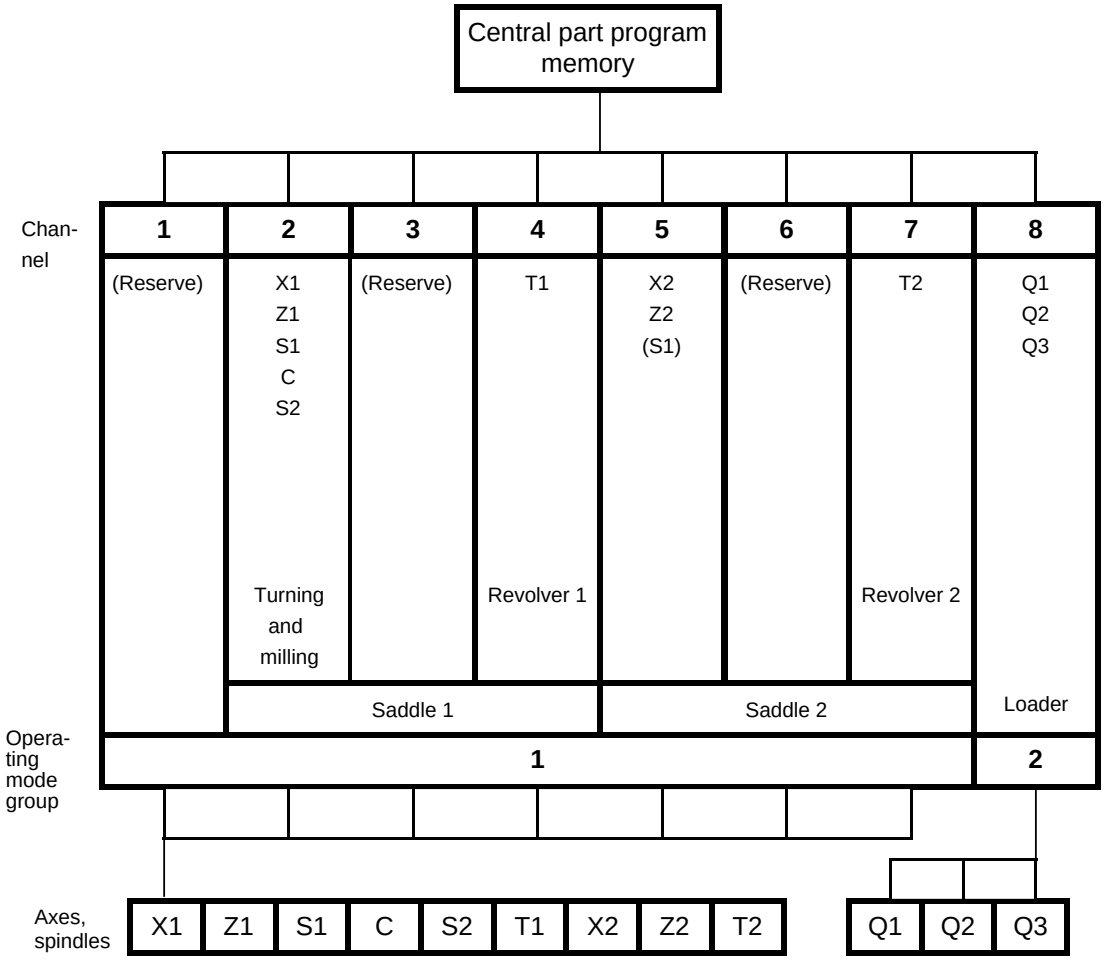
Channel structure implemented on twin saddle, single-spindle turning machine with milling capability, NC-controlled tool turrets and integral loader:



Schematic diagram of twin saddle, single-spindle turning machine for turning/milling with tool turret and integral loader

- Saddle 1, main spindle, auxiliary spindles (X_1, Z_1, S_1, S_2)
- Turret 1 (T_1)
- Saddle 2 (X_2, Z_2)
- Turret 2 (T_2)
- Loader (Q_1, Q_2, Q_3)

© Siemens AG 1991 All Rights Reserved



Channel structure: programmable channel assignment

A mode group contains the channels which must operate **simultaneously in the same operating mode** during machining.

Within the operating mode group, any axis can be programmed in any channel. In this example the loader has its own mode group. This makes it possible for the loader to be set up while workpieces are being produced.

In addition to the five channels in which control programs for the working units run, channels are reserved for future options (such as automatic tool retraction on tool breakage or for coordination programs).

In the **AUTOMATIC** mode basic display the program number is defaulted for each channel.

© Siemens AG 1991 All Rights Reserved

Selecting and deselecting Terminal Emulation VT340



This key only exists on the SINUMERIK 880, Software Version 6, if the Terminal Emulation option has been implemented. On SINUMERIK 880 GA2, Software Version 1, the key is implemented as standard.

This key is used to select and deselect Terminal Emulation.

Acknowledge alarm



With the operation of this key:

- you **acknowledge** the information from the NC diagnostics displayed on the second line of the CRT display
  - Alarm text
  - Alarm number

  for alarms marked with this key symbol in Section 4

- you **clear** the red LED ? on the LED panel



© Siemens AG 1991 All Rights Reserved

Actual position in double-height characters



When you operate this key, the CRT display of actual position is shown in double-height characters.

Operating the key again brings you back to the previous full CRT display (with standard size characters).

((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-0HC01-0BA0
Seite 2–17 einmontieren))

The distance to go is displayed in standard size characters.

The other information of the previous display is faded out.

Actual position on the display in double-height characters



If there are more than five axes, the 6th and further axes can be displayed by switching over to double-height characters and pressing the **page forwards** key (up to Software Version 6).

Please note when searching in a part program:

- Addresses (other than the address N ...) and words are only searched for from the current cursor position in the direction of the end of program.
- If the expression being searched for is between the start of the program and the current cursor position, it will not be marked;

  the "CHARACTER NOT FOUND" message is displayed in the operator prompt line.
- Block numbers (e.g. "N85") are searched for by the control and marked in both the direction of program end and program start.

© Siemens AG 1991 All Rights Reserved

Diagnostics and installation

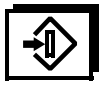


This switch is intended for:

- Installation
- Service

Please see the Installation Guide for SINUMERIK 880 for further information.

Search key (search for address**/**block no.**/**word**/**calling up data)



- You operate this key when you want to **search** in a **part program** for:
  - an address
  - a block number
  - a word
- **or** when you want to **display on the screen**:
  - a tool compensation number (with the appropriate data)
  - machine data (MD) or setting data (SD).

  Before operating the Search key you must write the data to be searched for into the input line:

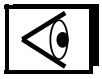
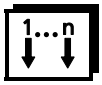
  Upon operation of the key, the cursor (appearance**/**format ▮) is positioned immediately **before** the term searched for.
- Please note when searching in a part program:
  - Addresses (other than the address N...) and words are only searched for from the current cursor position in the direction of the end of program.
  - If the data sought is between the start of the program and the current cursor position, it will not be found; the "CHARACTER NOT FOUND" message is shown in the operator prompt line.
  - Block numbers (e.g. "N85") are searched for by the control and marked in both the direction of program end and program start.
- To display the actual values as from a certain axis number, you must enter the axis number in the input field and then press the search key (only possible in JOG, INC and AUTOMATIC modes).

© Siemens AG 1991 All Rights Reserved

2.1.1.10 Assignment of the full standard keyboard in NC mode

The full standard keyboard is available as an option.

In NC mode you can carry out NC operations on the full standard keyboard. Several SINUMERIK operator functions have been transferred to the full standard keyboard for this reason. The following keys have been transferred to the full standard keyboard:

- 5 softkeys including the ETC and RECALL keys
- Control keys for
  - Diagnostics and installation 
  - Acknowledge alarm 
  - Channel switchover 
  - Actual position is double-height characters 
  - Selection/deselection of Terminal Emulation (option) 
- Editing and input keys
  - Modify word 
  - Search key 
  - Delete word/block 
  - Block end (LINE FEED) 

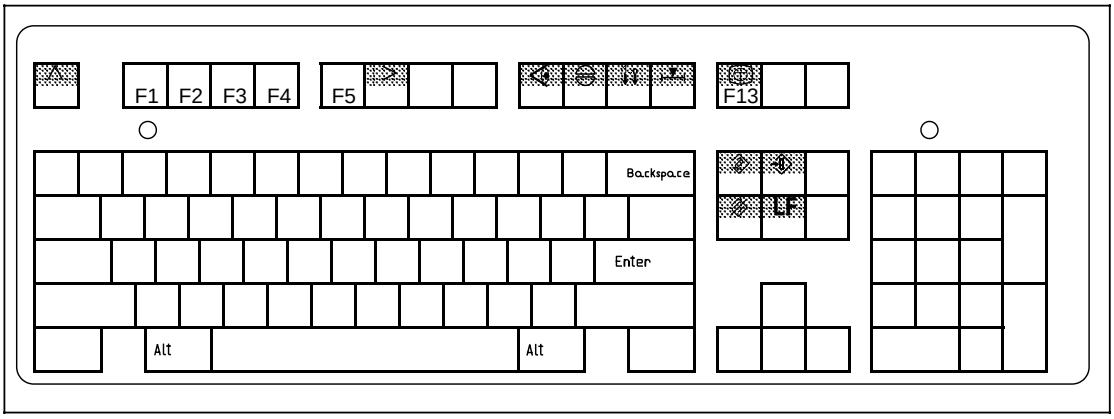
The key symbols for these key functions are supplied with the full standard keyboard. Replace the key caps in question to obtain a SINUMERIK-specific full standard keyboard.

The remaining keys on the full standard keyboard are enabled according to the standard operator panel assignment. The input key corresponds to the enter key on the full standard keyboard. Similarly, the delete or clear key corresponds to the backspace key.

© Siemens AG 1991 All Rights Reserved

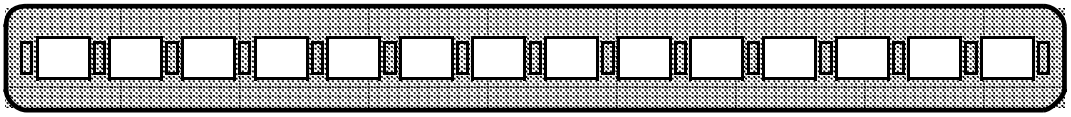
Note:

The customer keys and the mode group selection keys are not shown on the full standard keyboard.

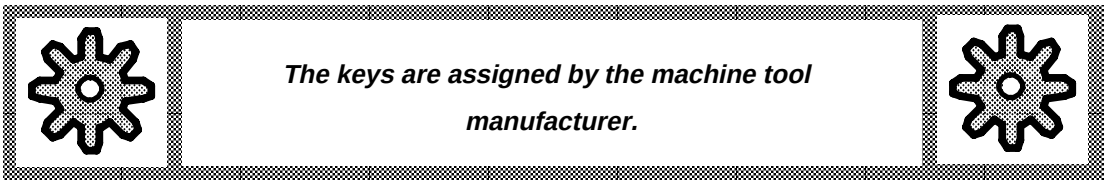


Full standard keyboard assignment in NC mode

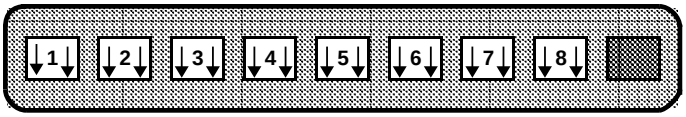
2.1.1.11 User-assignable keys



View of the user-assignable keys (under the softkeys)



2.1.1.12 Operating mode group selection keys



View of the operating mode group selection keys

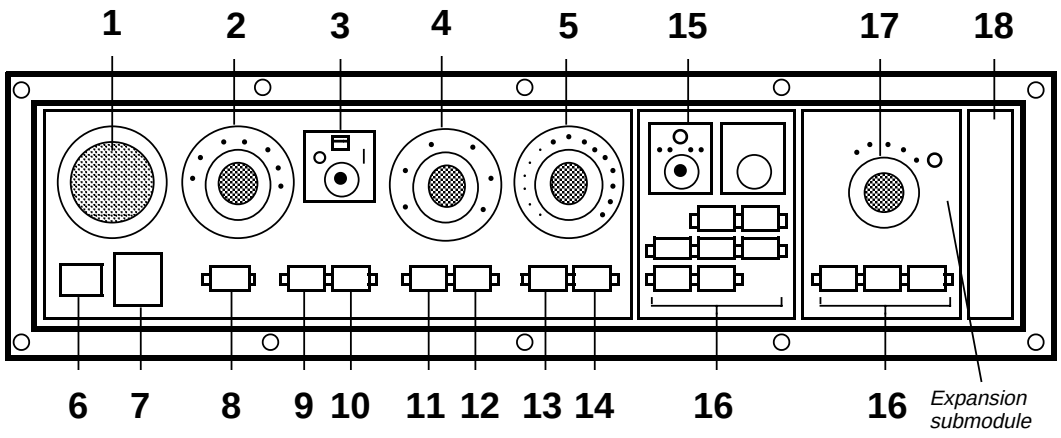
The operating mode groups combine NC channels. Which channels are combined in your control is fixed in machine data.

© Siemens AG 1991 All Rights Reserved

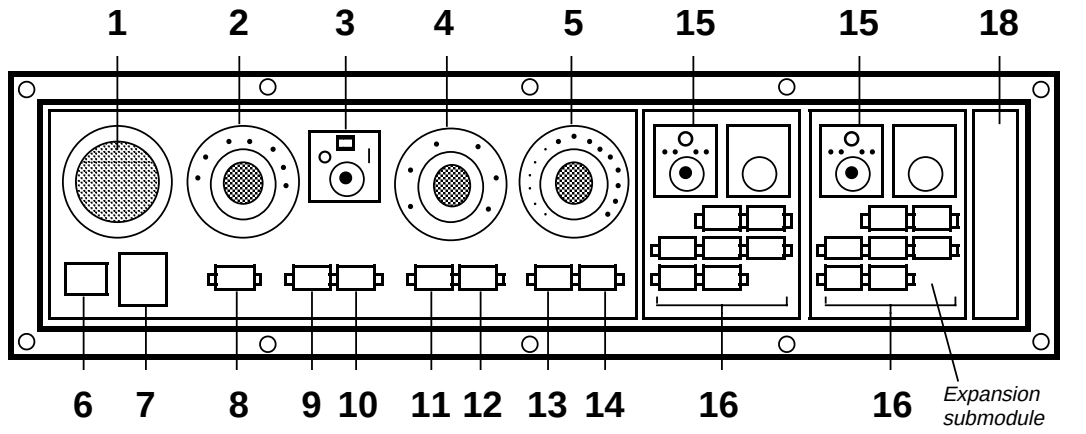
2.1.2 Operator interface of the machine control panel for SINUMERIK 880

Versions for SINUMERIK 880T:

Machine control panel with AUX. AXES expansion submodule



Machine control panel with DOUBLE SADDLE expansion submodule

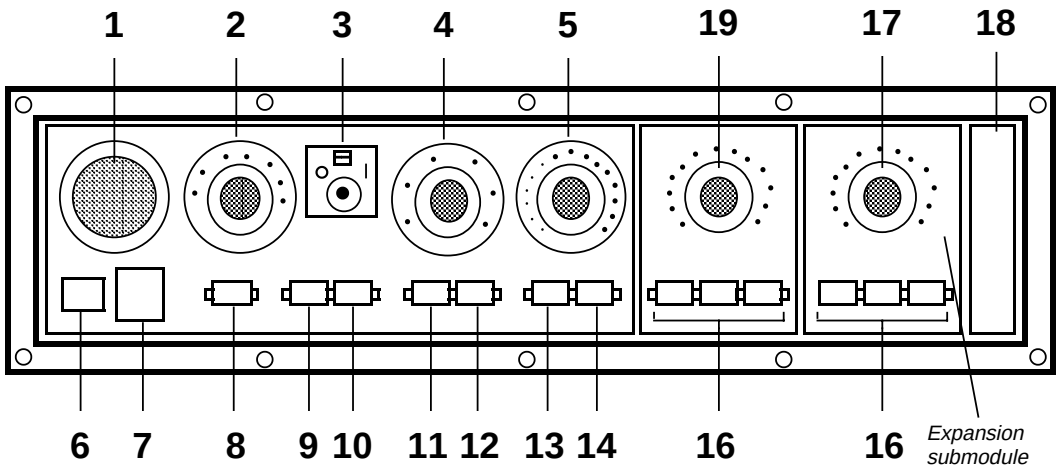


1 Emergency Stop switch
2 Operating mode selector switch
3 Single block switchover
4 Spindle speed override switch
5 Feedrate/rapid traverse overr. switch
6 NC-ON key
7 Key-operated switch
8 RESET key
9 "Program Stop" (NC-STOP) key
10 "Program Start" (NC-START) key
11 Spindle stop key
12 Spindle start key
13 Feed hold key
14 Feed start key
15 Handwheel selector switch
16 Direction keys, rapid traverse key
17 Auxiliary axes selector switch
18 Serial interface
19 Axis selector switch (880M)

© Siemens AG 1991 All Rights Reserved

Version for SINUMERIK 880M:

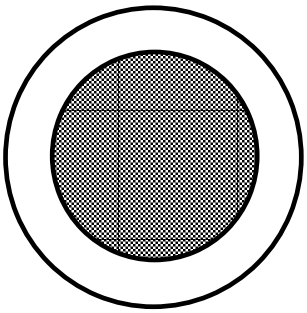
Machine control panel with AUX. AXES expansion submodule



(See previous page for Legend)

Explanation of the operating elements

Emergency stop switch



You operate the red switch in **emergency situations**:

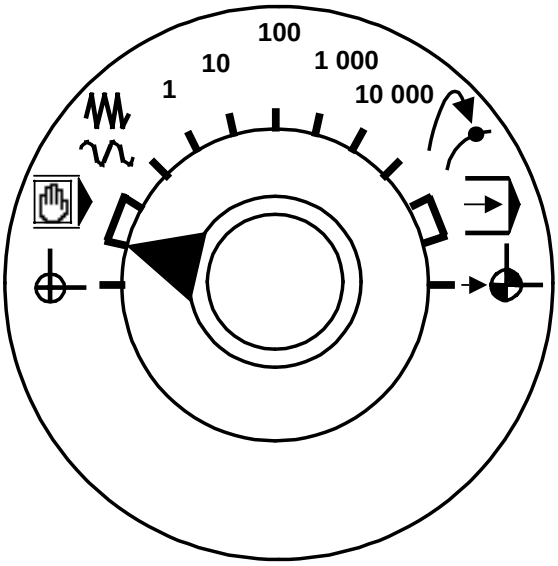
- when there is a danger to life
- when there is a danger that the machine or workpiece could be damaged.

Operation of the ”Emergency stop” switch generally brakes all drives with maximum braking power to a defined state.

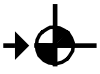
For further or other reactions to "Emergency stop":
See machine tool manufacturer's documentation.

© Siemens AG 1991 All Rights Reserved

Operating mode selector switch

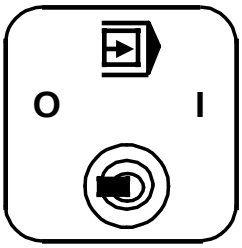


This rotary switch with 13 latched positions enables you to select the following operating modes:

Symbol on the selector switch	Function	Designation of the operating mode
	**SET ACTUAL VALUE**	**PRESET** Preset Setpoint (1st position)
	**MANUAL DATA INPUT/AUTOMATIC**	**MDI-AUTOMATIC** Manual Data Input - Automatic (2nd and 3rd position)
	**FEED/JOG**	**JOG** Jogging (4th position)
1, 10, 100, 1 000, 10 000	**INCREMENTAL TRAVERSE**	**INC FEED...** Incremental Feed (5th to 9th position)
	**REPOSITIONING** Return to contour	**REPOS** Reposition (10th position)
	**AUTOMATIC OPERATION** Program processing	**AUTOMATIC** (11th and 12th position)
	**APPROACH REFERENCE POINT**	**REFPOINT** Reference Point (13th position)

© Siemens AG 1991 All Rights Reserved

Single block switchover



This switch enables you to run a part program on a block-by-block basis in the **AUTOMATIC** operating mode.

When the switch is in pos. **"0"**: Single block operat. is not active!

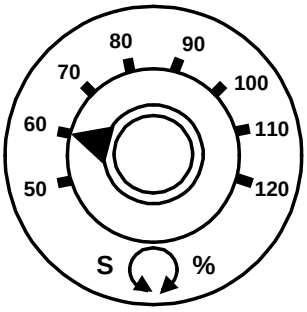
When the switch is in pos. **"1"**: Single block operation is active!

When single block operation is active:

- the "SBL" (Single block) message is shown on the first line of the CRT
- the current part program block is only processed when you press the NC START key
- when the block has been processed, the ”HOLD SINGLE BLOCK” message is displayed on the CRT
- when you operate the NC START key again, the next block is called and processed etc...

For certain functions (e.g. coordinate rotation, soft approach to contour) the control inserts further blocks. Depending on the number of blocks inserted, the NC START key must be pressed several times.

Spindle speed override switch



- The rotary switch, with 16 latched positions, enables you to decrease or increase the programmed spindle speed "S" (relative to 100%).
- The actual function of the switch depends on machine data.
- The set spindle speed value "S" is displayed on the CRT as an absolute value and as a percentage.

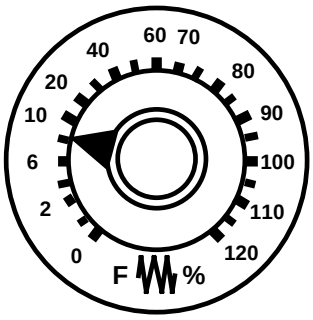
Control range: 50% to 120% of the programmed spindle speed

Increment: 5% from position to position

Note: The given increment and control range are valid for standard machine data (MD). This MD is user-specific and can be altered by the machine tool manufacturer.

© Siemens AG 1991 All Rights Reserved

Feed/rapid override switch



The rotary switch, with 23 latched positions, enables you to decrease or increase the **programmed** feedrate value "F" (relative to 100%).

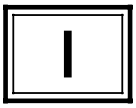
The set feedrate value "F" is displayed on the CRT in %.

Control range: 0% to 120% of the programmed feedrate. In **rapid traverse**, the 100% value is not exceeded.

Step value: 0 %, 1 %, 2 %, 4 %, 6 %, 8 %, 10 %, 20 %, 30 %, 40 %, 50 %, 60 %, 70 %, 75 %, 80 %, 85 %, 90 %, 95 %, 100 %, 105 %, 110 %, 115 %, 120 %

Note: The given increment and control range are valid for standard machine data. This MD is user-specific and may be altered by the machine tool manufacturer.

Switch for switching on the NC control



By operating this key, you switch the NC control on.

© Siemens AG 1991 All Rights Reserved

Key-operated switch for input inhibit/operation inhibit



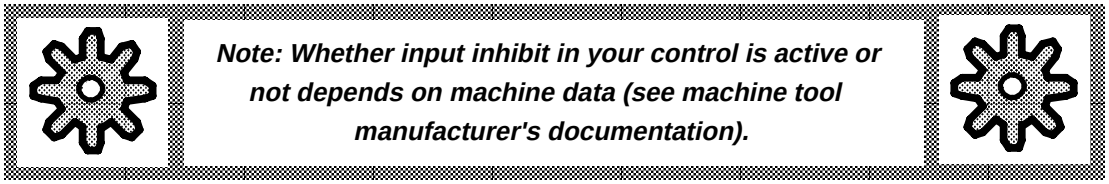
You can inhibit data input with the key-operated switch.
Associated operating functions are then no longer possible.



The key can be withdrawn. In this position the input or operation inhibit is active.



The key **cannot** be withdrawn. Input or operation inhibit is cancelled.



Note: Whether input inhibit in your control is active or not depends on machine data (see machine tool manufacturer's documentation).

RESET



You operate the RESET key:

- The current part program is interrupted.
- Alarms and diagnostics messages are cleared (RESET alarms)
- The control is switched to the Reset state:
  – the NC remains synchronised with the machine
  – all buffer and working memories are cleared (the part program memory remains unchanged)
  – the control is in the reset condition and ready for a new program start.

Program stop/program start (NC STOP/NC START)



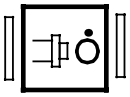
You press the NC STOP key:
The part program being processed is interrupted. You can continue operation by pressing NC START.



You press the NC START key:
The part program called is started at the current block. In automatic operation the overstored functions are transferred to the PLC.

© Siemens AG 1991 All Rights Reserved

Spindle stop/Spindle start

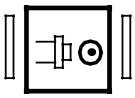


You operate the "Spindle stop"key:

- the spindle is brought to a standstill

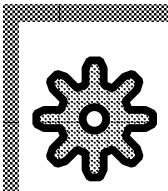
Example for the use of "Spindle stop"

- during operation in **MDI-AUTOMATIC** mode, a block with a fault is discovered
- in **JOG, INC.., REPOS** modes, e.g. during return to the contour
- to change a tool
- to input S, T, H, M functions during setting up (overstore)



You operate the "Spindle start" key:

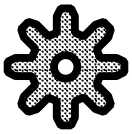
- the spindle speed accelerates to the value specified in the program



The following values are specified in machine data:

- ***the max. spindle speed***
- ***the values for the spindle speed override switch positions***

(see the Machine Tool Manufacturer's Documentation)



Feed hold/Feed start



You operate the "Feed hold" key:

- the program being processed is stopped
- the feed drives are brought to a standstill
- the red “Feed hold” LED lights up

Example for the use of "Feed hold"

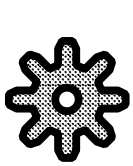
- during operation in **MDI-AUTOMATIC** mode, a block with a fault is discovered
- in **JOG, INC.., REPOS** modes, e.g. during return to the contour
- to change a tool
- to input S, T, H, M functions during setting up (overstore)

© Siemens AG 1991 All Rights Reserved



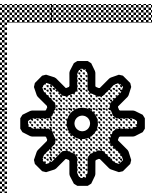
You operate the "Feed start" key:

- the part program continues in the current block
- the feedrate is increased to the value given in the program

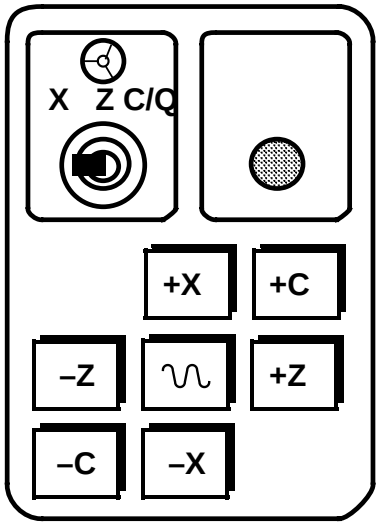


The following values are specified in machine data:

- ***the feed and rapid traverse rates***
- ***the values for the feedrate override switch positions***
- ***whether the feedrate override switch is also active for rapid traverse***

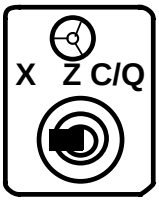


Handwheel selection and direction keys/Jogging axes (T version only)



The toggle switch for handwheel selection and the direction keys enable you to jog the axes in the **JOG, REPOS** or **INC...** modes.

Functions of the Handwheel selection toggle switch



With this toggle switch you traverse the desired axis:

"left" position:	With this key you traverse the "X" axis
"centre" position:	With this key you traverse the "Z" axis
"right" position:	With this key you traverse the "C/Q" axis (auxiliary axis)

© Siemens AG 1991 All Rights Reserved

Function of the direction keys

+X	With this key you traverse the "X" axis	Only when the toggle switch is in the "X" position
–X	With this key you traverse the "X" axis in the opposite direction	
+Z	With this key you traverse the "Z" axis	Only when the toggle switch is in the "Z" position
–Z	With this key you traverse the "Z" axis in the opposite direction	
+C	With this key you traverse the "C" or "Q" axis	Only when the toggle switch is in the "C/Q" position
–C	With this key you traverse the "C" or "Q" axis in the opposite direction	
∿∿	Rapid traverse overlay: When you operate this key at the same time as any of the keys above, the axis is traversed in rapid traverse mode.	

- The "Feed hold" LED must not be on
- The CRT shows you the **specified** feedrate value "F" at which the axes will travel when you operate the direction keys.
  The value is displayed as an absolute value and as a "%" of the **programmed** feedrate "F" (see: "Feedrate**/**rapid traverse override switch" in this section).
- In **JOG** the feed motion is arbitrary. The traverse path is only limited by the end limit switch.
- In **REPOS** the feed motion is arbitrary (see **JOG**).
  If, however, the point of interruption (in a part program that has been partially run) is reached, the direction keys become **inactive**.
- The direction keys can traverse the axes: – in **continuous mode**
  – in **jog operation**

Continuous mode:

When the direction key is pressed (whether you press for a short or long time) the axis only traverses one increment (1**/**10**/**100**/**1000**/**10000 increments of display resolution).

© Siemens AG 1991 All Rights Reserved

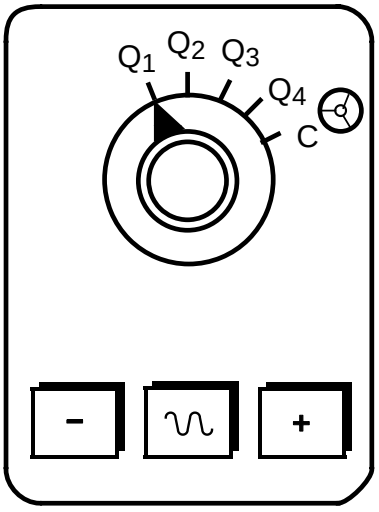
Jog operation:

The axis will traverse as long as you press the direction key. Traversing stops when you release the key. This also happens if the set increment has not been reached.

The machine tool manufacturer specifies with machine data:

- ***whether traversing with the direction keys is in:***
  - ***continuous mode or***
  - ***jog mode***
- ***the traversing speed in "rapid traverse"***

Axis selector switch for auxiliary axes and direction keys



This submodule allows you to jog further auxiliary axes.

Function of the axis selector switch

You select the axis to be traversed with this rotary switch:

In the example auxiliary axis "Q_1".

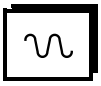
Function of the direction keys



You traverse the axis selected.



You traverse the axis selected in the opposite direction.



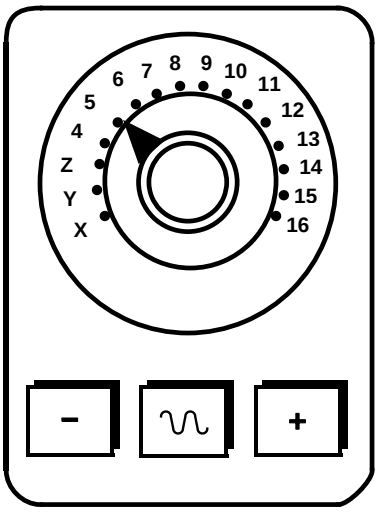
Rapid traverse:

If you operate this key **simultaneously** with one of the above-mentioned keys, the axis is traversed in **rapid traverse** mode.

The axis selector switch for auxiliary axes is only available with Option J83.

© Siemens AG 1991 All Rights Reserved

Axis selector switch for direction keys/jogging axes



The axis selector switch and direction keys enable you to jog the axes in the **JOG, REPOS** or **INC...** modes.

- The "Feed hold" LED must not be on
- The CRT shows you the **specified** feedrate value "F" at which the axes will travel when you operate the direction key. The value is displayed as an absolute value and as a "%" of the **programmed** feedrate "F"(see: "Feedrate**/**rapid traverse override switch" in this section)
- In **JOG** the feed motion is arbitrary. The traverse path is only limited by the end limit switch.
- In **REPOS** the feed motion is arbitrary (see **JOG**). If, however, the point of interruption (in a part program that has been partially run) is reached, the direction keys become **inactive**.
- The direction keys can traverse the axes:
  – in **continuous mode**
  – in **jog operation**

Continuous mode:

When the direction key is pressed (whether you press for a short or long time) the axis only traverses one increment (1**/**10**/**100**/**1000**/**10000 increments of display resolution).

Jog operation:

The axis will traverse as long as you press the direction key. Traversing stops when you release the key. This also happens if the set increment has not been reached.

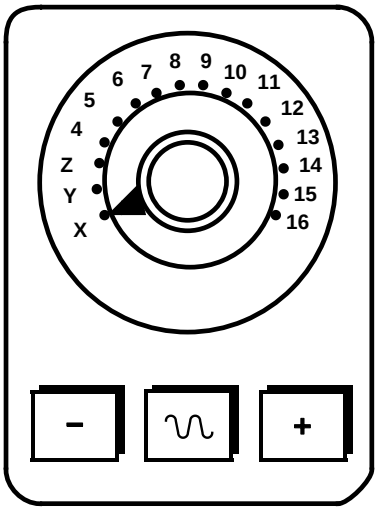
It is specified by means of machine data whether traversing with the direction keys is in:

- ***continuous mode or***
- ***jog mode***

(see machine tool manufacturer's documentation)

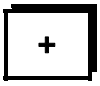
© Siemens AG 1991 All Rights Reserved

Function of the axis selector switch



You select the axis to be traversed with the axis selector switch, (in example shown: axis 1).

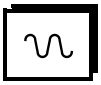
Function of the direction keys



You traverse the axis selected.



You traverse the axis selected in the opposite direction.



Rapid traverse:
If you operate this key **simultaneously** with one of the above-mentioned keys, the axis is traversed in **rapid traverse** mode.

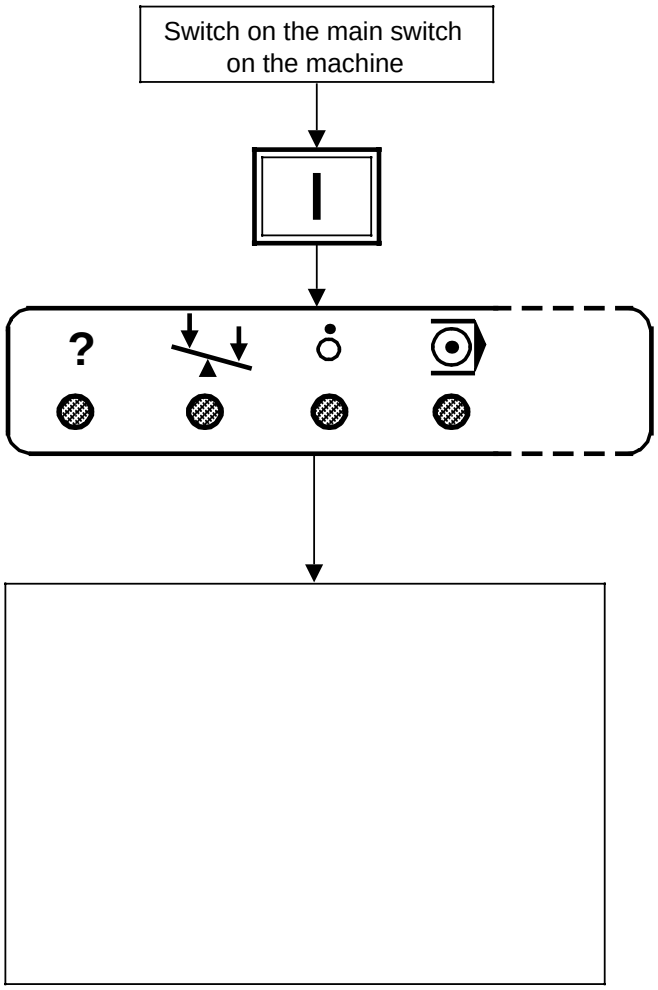
The traversing speed in "rapid traverse" is specified in machine data

(see machine tool manufacturer's documentation)

© Siemens AG 1991 All Rights Reserved

2.2 Switching the control on/off

2.2.1 Switching on the SINUMERIK 880



Now operate the "NC ON" key on the machine control panel

Reaction of the display panel on the control:

For approx. 3 seconds the 4 LEDs are lit and then extinguished.

Approx. 10 seconds after switching on, you will see a display on the CRT in the mode selected with the mode selector switch, e.g. **JOG** mode, similar to the example shown here (e.g. Program "%77").

The control is switched on.

2.2.2 Switching off the SINUMERIK 880

You switch the control off with the main switch on the machine. All axes and spindles must have been brought to the stop state.

© Siemens AG 1991 All Rights Reserved

2.3 Operating modes

2.3.1 General

On a machine tool an NC controls according to a specified program – the **part program**:

- the motion of the tool
- the motion of the workpiece

Beyond this, further preparations are required with a numerically controlled machine tool before the actual machining process can be started. For these preparations, the control has to be set to certain **operating states** to prepare it for certain operations.

These include:

- Traversing the tools or the workpiece to the start position required in the setting up plan
- Loading the part program into the memory of the control
- Checking and entering the zero offsets
- Checking and entering the tool offsets

The SINUMERIK 880 has **7 operating modes** to bring the control into the desired state.

2.3.2 Operating modes - overview

- **Automatic operation** (CRT display: **AUTOMATIC**)

To process a part program in this operating mode, the control calls the blocks in sequence and evaluates them. The evaluation takes all the relevant offsets into account. The blocks prepared in this way are processed in sequence.

The part program can be entered in the control via the universal interface (e.g. via punched tape), a computer link or via the keyboard.

While one part program is being processed, another part program can be entered simultaneously.

((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-0HC01-0BA0
Seite 2–33 einmontieren))

***AUTOMATIC** basic display*

© Siemens AG 1991 All Rights Reserved

- **Feed/Jog** (CRT display: **JOG**)

With the direction keys and the preset feedrate value "F", you can traverse the tool as desired.

After a program interruption, you can see the distance to the point of interruption displayed in the "REPOS offset".

You traverse to the point of interruption until the REPOS offset shows zero.

((Foto aus der Druckschrift SINUMERIK 880, Bedienungsanleitung, Bestellnr. 6ZB5 410-0HC01-0BA0 Seite 2–34 oben einmontieren))

***JOG** basic display*

- **Manual Data Input/Automatic** (CRT display: **MDI AUTOMATIC**)

In this operating mode, you can input part program blocks into the buffer memory of the control.

The control processes the input block, and then clears the buffer memory ready for new data.

Used, for example, in connection with operations in **JOG** or **INC FEED** modes.

((Foto aus der Druckschrift SINUMERIK 880, Bedienungsanleitung, Bestellnr. 6ZB5 410-0HC01-0BA0 Seite 2–34 mitte einmontieren))

***MDI AUTOMATIC** basic display*

- **Approach to reference point** (CRT display: **REFPOINT**)

The control and machine are synchronized in this mode after switching on.

The direction keys are used to move the machine in either jog or continuous operation, depending on the machine data set.

The reference point must be approached in each axis individually.

When the reference point has been reached, the actual value memory is set to the value of the reference point coordinates.

((Foto aus der Druckschrift SINUMERIK 880, Bedienungsanleitung, Bestellnr. 6ZB5 410-0HC01-0BA0 Seite 2–34 unten einmontieren))

***REFPOINT** basic display*

© Siemens AG 1991 All Rights Reserved

- **Incremental 1...10 000 feed** (CRT display: **INC FEED 1...10 000**)

In this operating mode, defined paraxial positioning is possible using the direction keys. The feedrate is fixed with machine data. Feedrate override (in the range 0% ... 120%) is only possible when the appropriate interface signals are transferred from the PLC to the NC.

MD define whether the set increment will be traversed in continuous mode or jog operation.

((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-0HC01-0BA0
Seite 2–35 oben einmontieren))

INC FEED 1 ... 10 000 *basic display*

- **Set actual value** (CRT display: **PRESET**)

The directions of movement of an NC machine are based on a right-angled system of coordinates assigned to the machine axes. In the absolute machine coordinate system, control zero can be offset with reference to the machine zero point.

In **PRESET** mode, the control zero point can be placed anywhere within the machine coordinate system.

((Mitgeliefertes Foto
Nr. 3–132 einmontieren))

PRESET *basic display*

- **Repositioning** (CRT display: **REPOS**)

In **REPOS** mode, the tool can be returned to the point of interruption using the direction keys and the set feedrate value ”F”.

The REPOS offset display shows the distance from the actual position to the point of interruption with the correct sign.

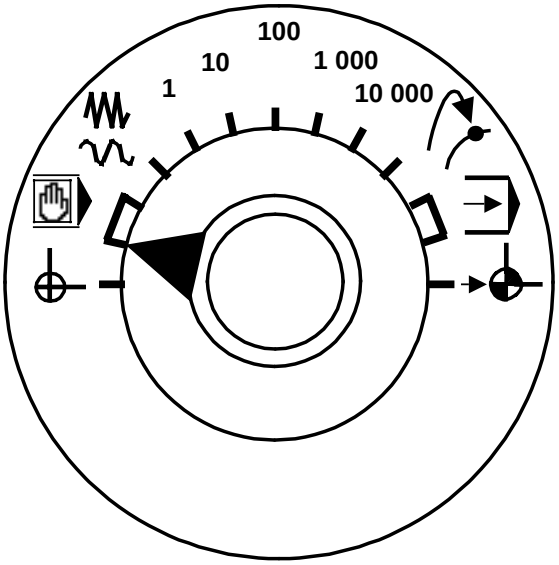
When the point of interruption is reached, the REPOS offset display is zero; at the same time the direction keys are no longer active.

((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-0HC01-0BA0
Seite 2–35 unten einmontieren))

REPOS *basic display*

© Siemens AG 1991 All Rights Reserved

2.3.3 Selection of operating modes with external machine control panel



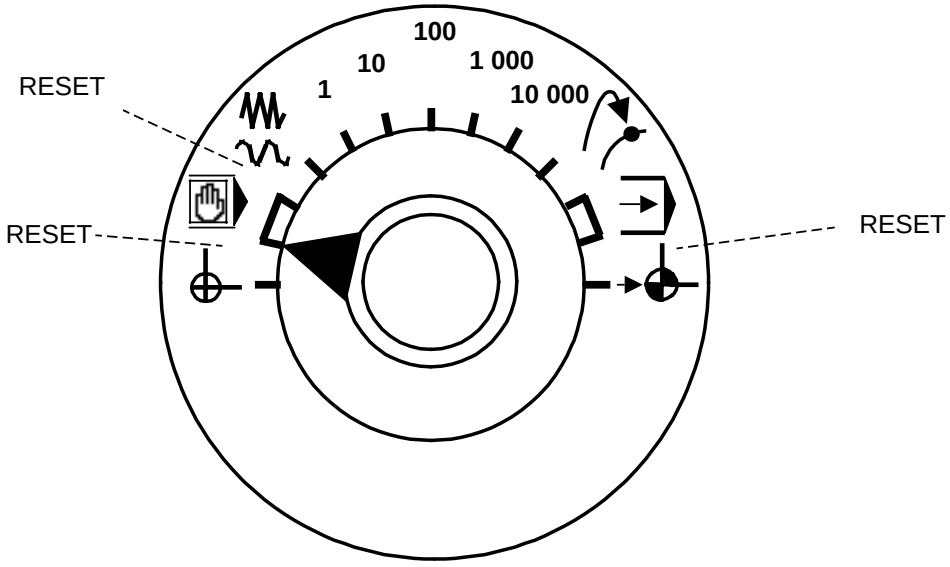
You rotate the operating mode selector switch to the desired position.

Symbol on the selector switch	Function	Designation of the operating mode
	**SET ACTUAL VALUE**	**PRESET** Preset Setpoint (1st position)
	**MANUAL DATA INPUT/ AUTOMATIC**	**MDI-AUTOMATIC** Manual Data Input-Automatic (2nd and 3rd position)
	**FEED/JOG**	**JOG** Jogging (4th position)
1, 10, 100, 1 000, 10 000	**INCREMENTAL FEED**	**INC FEED...** Incremental Feed (5th to 9th position)
	**REPOSITIONING** Return to contour	**REPOS** Reposition (10th position)
	**AUTOMATIC OPERATION** Program processing	**AUTOMATIC** (11th and 12th position)
	**APPROACH REFERENCE POINT**	**REFPOINT** Reference Point (13th position)

© Siemens AG 1991 All Rights Reserved

2.3.4 "RESET" with change of operating mode

- When changing from one selected operating mode to another, a RESET is generated by the control.
- The RESET generated by the control when changing has the same effect as if the RESET key had been operated.
- The generation of a RESET depends on the modes switched.



Examples:

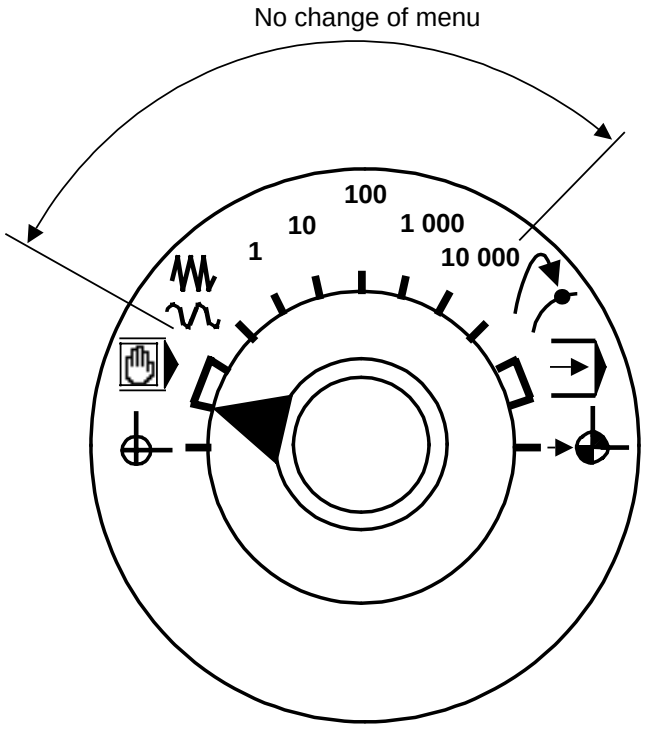
- When changing from  (**AUTOMATIC**) to  (**JOG**) mode, **no** "Reset" is generated by the control!

- When changing from  (**JOG**) to  (**REFPOINT**) mode, a "Reset" is generated by the control!

© Siemens AG 1991 All Rights Reserved

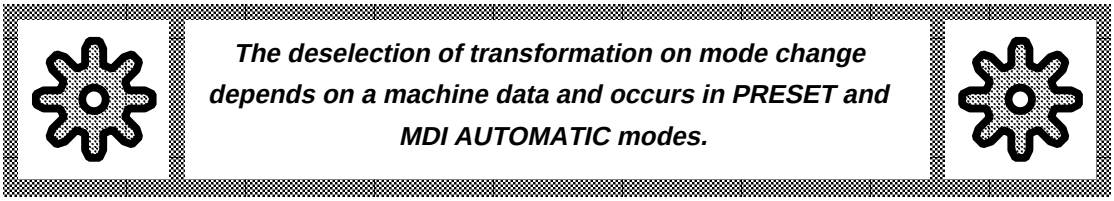
2.3.4.1 No change of menu when switching between JOG/INC

When switching between the **JOG** and **incremental feed** operating modes, there is no jump into the basic menu; the selected menu remains.



© Siemens AG 1991 All Rights Reserved

2.3.4.2 Transformation states in the modes



Mode	Explanation
PRESET MDI AUTOMATIC	Transformation deselection depending on a machine data
JOG INC FEED REPOS AUTOMATIC	Transformation maintained
REFPOINT	Transformation deselection

© Siemens AG 1991 All Rights Reserved

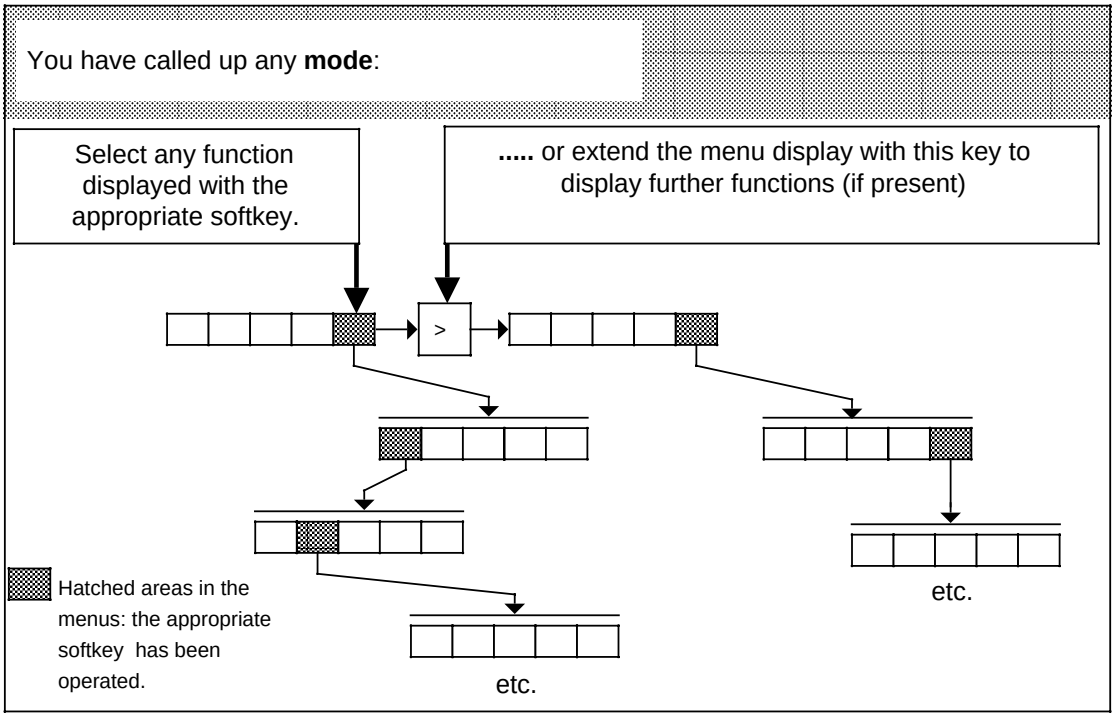
2.3.5 Branching to functions within an operating mode

Depending on the mode you have selected, you will be offered a "Basic softkey menu" of operating functions in the **menu display** on the CRT.

After you select a function using the softkeys, the control will display other/new menus.

Menu trees with several branches appear.

Menu trees for each operating mode are stored in the control:



Branching of the operating functions (menu tree) with stylized representation of the menus

Note :

The detailed branching structure is explained **separately for each operating mode** in Sections 2.4.1 to 2.4.9.

© Siemens AG 1991 All Rights Reserved

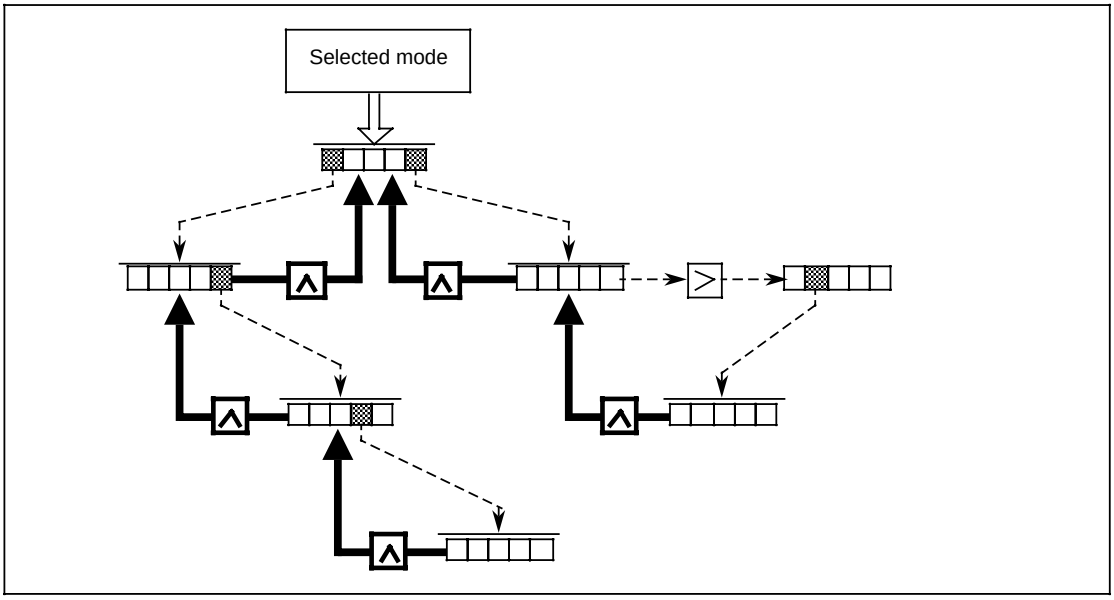
2.3.6 Jumping back to functions in higher-level menus within an operating mode

You wish to return to higher-level menus after repeated branching:

To do this, use the RECALL key:



If you operate this key once, the next menu up is displayed with operating functions.



Jumping back to operating functions in higher-level menus (black arrows), with stylized representation of menus

© Siemens AG 1991 All Rights Reserved

2.4 Operating mode menu trees

In this section you will find graphical overviews for **all** the SINUMERIK 880 **operating modes**.

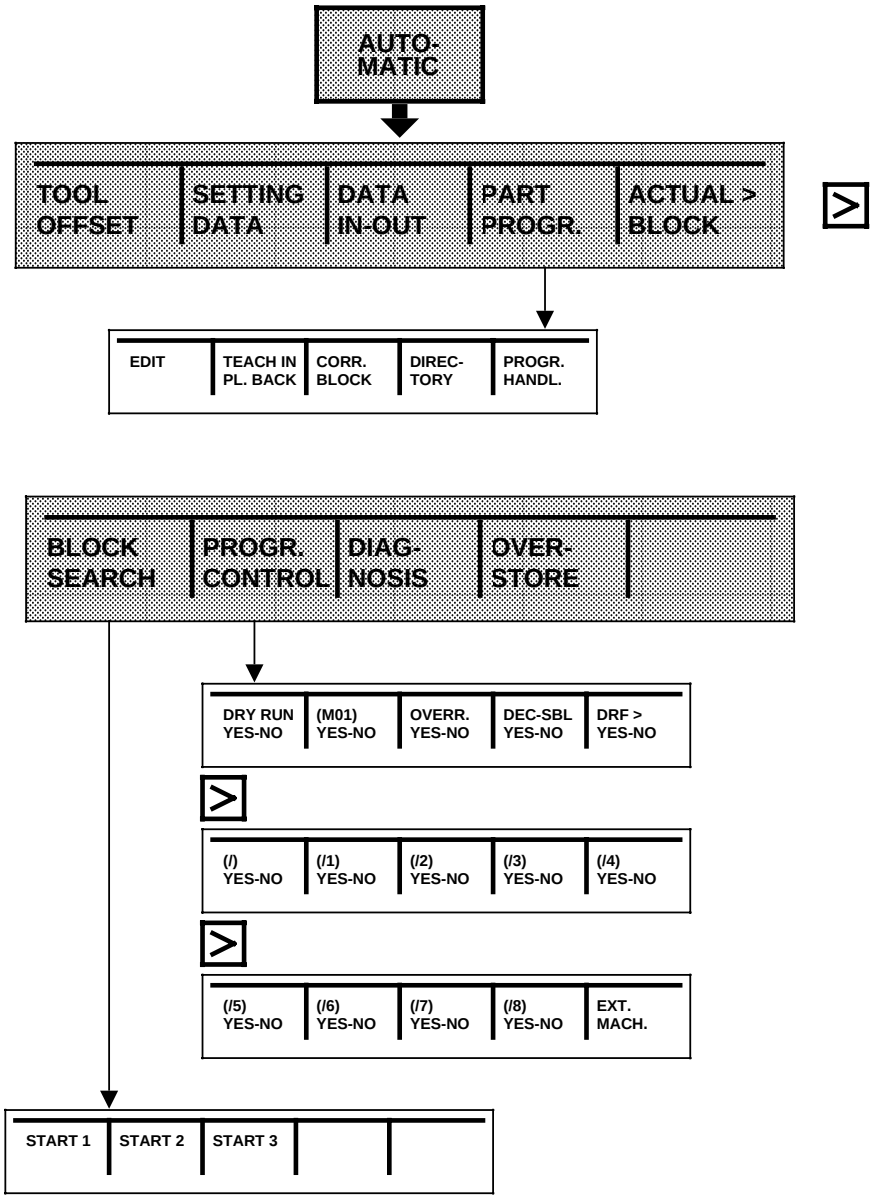
Terms and abbreviations used are explained in Section 2.5 ”Software function dictionary”.

For each operating mode all the appropriate branches to further functions are represented for. Functions common to all modes are described in Section 2.4.9 ”Operating functions”.

- **AUTOMATIC** mode
- **JOG** mode
- **JOG** mode
  (after **”**TEACH IN**/**PLAYBACK**”** function in AUTOMATIC mode)
- **MDI-AUTOMATIC** mode
- **PRESET** mode
- **INC FEED 1...10 000** mode
- **REFPOINT** mode
- **REPOS** mode
- Functions
  - SETTING DATA
  - DATA IN-OUT
  - PART PROGRAM
  - DIAGNOSTICS
  - OPERATOR GUIDANCE
    * CONTOUR
    * CYCLES (SINUMERIK 880T**/**880TT)
    * CYCLES (SINUMERIK 880M)
  - SIMULATION
    * SIMULATION DATA (SINUMERIK 880T**/**880TT)
    * SIMULATION DATA (SINUMERIK 880M)

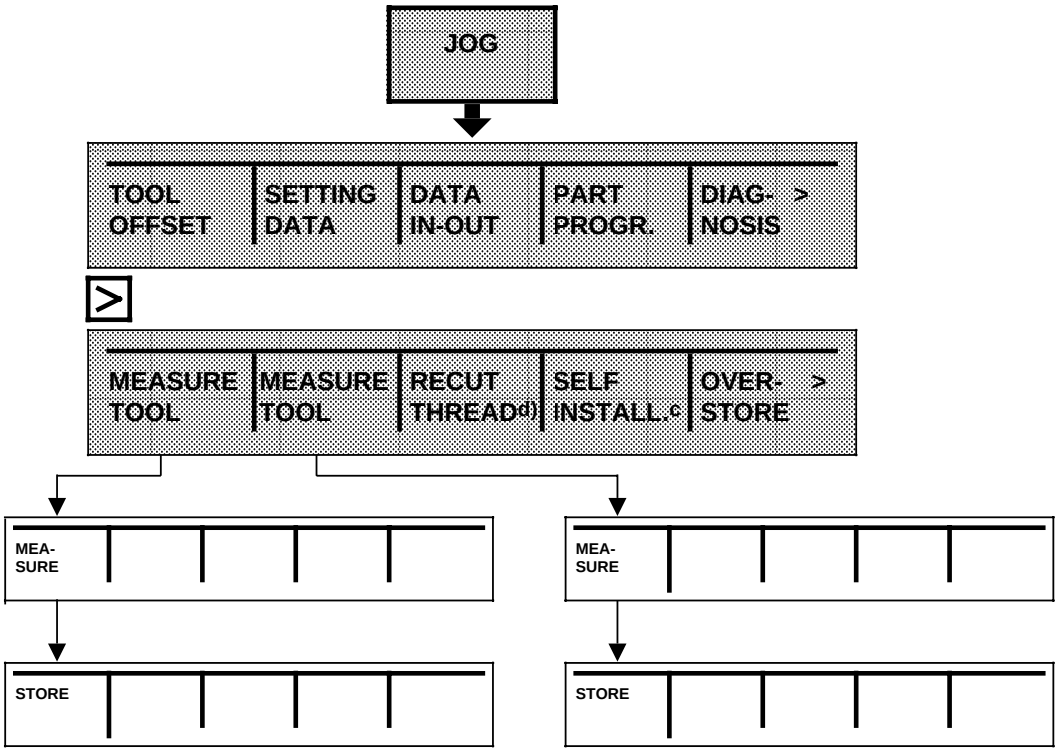
© Siemens AG 1991 All Rights Reserved

2.4.1 AUTOMATIC mode

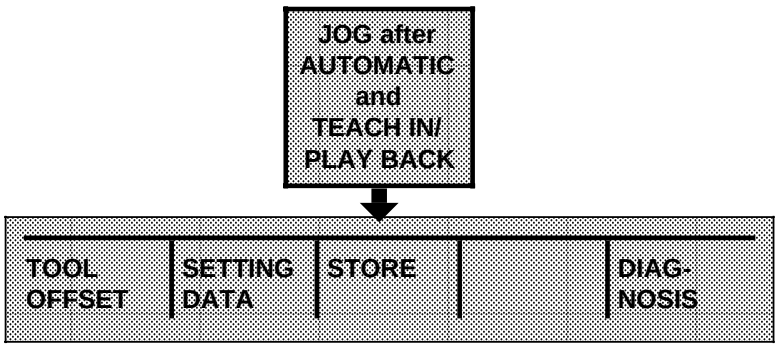


© Siemens AG 1991 All Rights Reserved

2.4.2 JOG mode



2.4.3 JOG mode (after TEACH IN/PLAYBACK in AUTOMATIC mode)

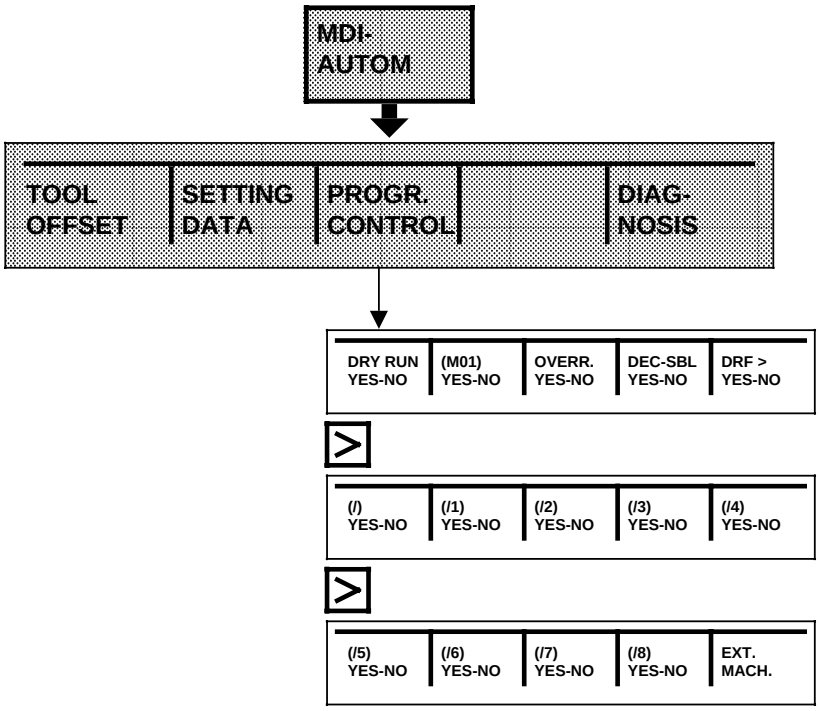


c) up to and including Software Version 6

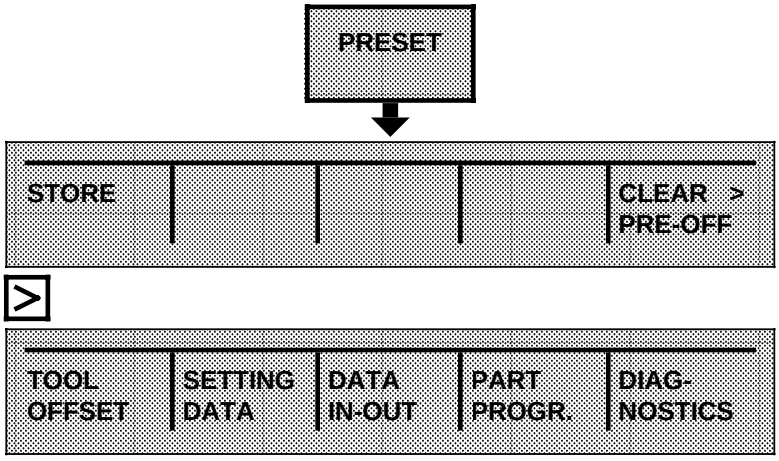
d) from GA2, Software Version 1

© Siemens AG 1991 All Rights Reserved

2.4.4 MDI-AUTOMATIC mode

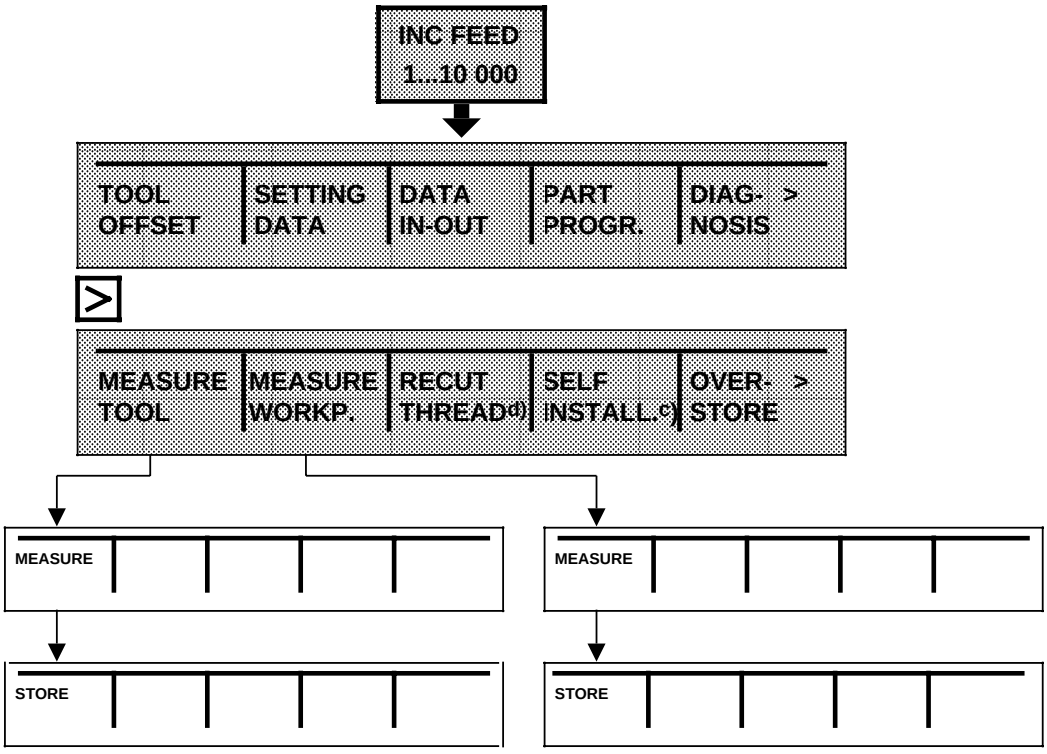


2.4.5 PRESET mode

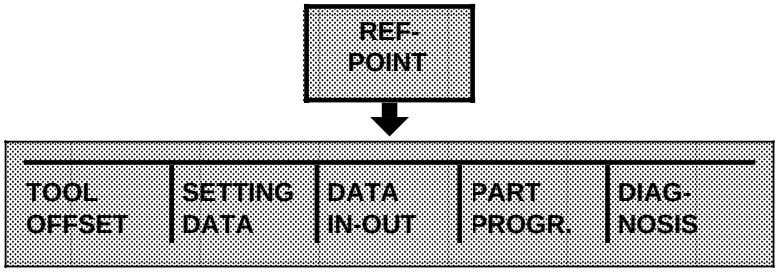


© Siemens AG 1991 All Rights Reserved

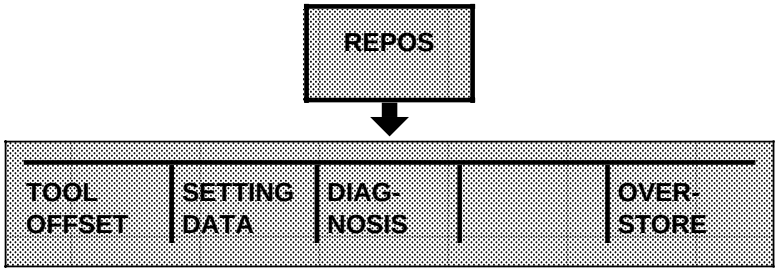
2.4.6 INC FEED 1...10 000 mode



2.4.7 REFPOINT mode



2.4.8 REPOS mode



---

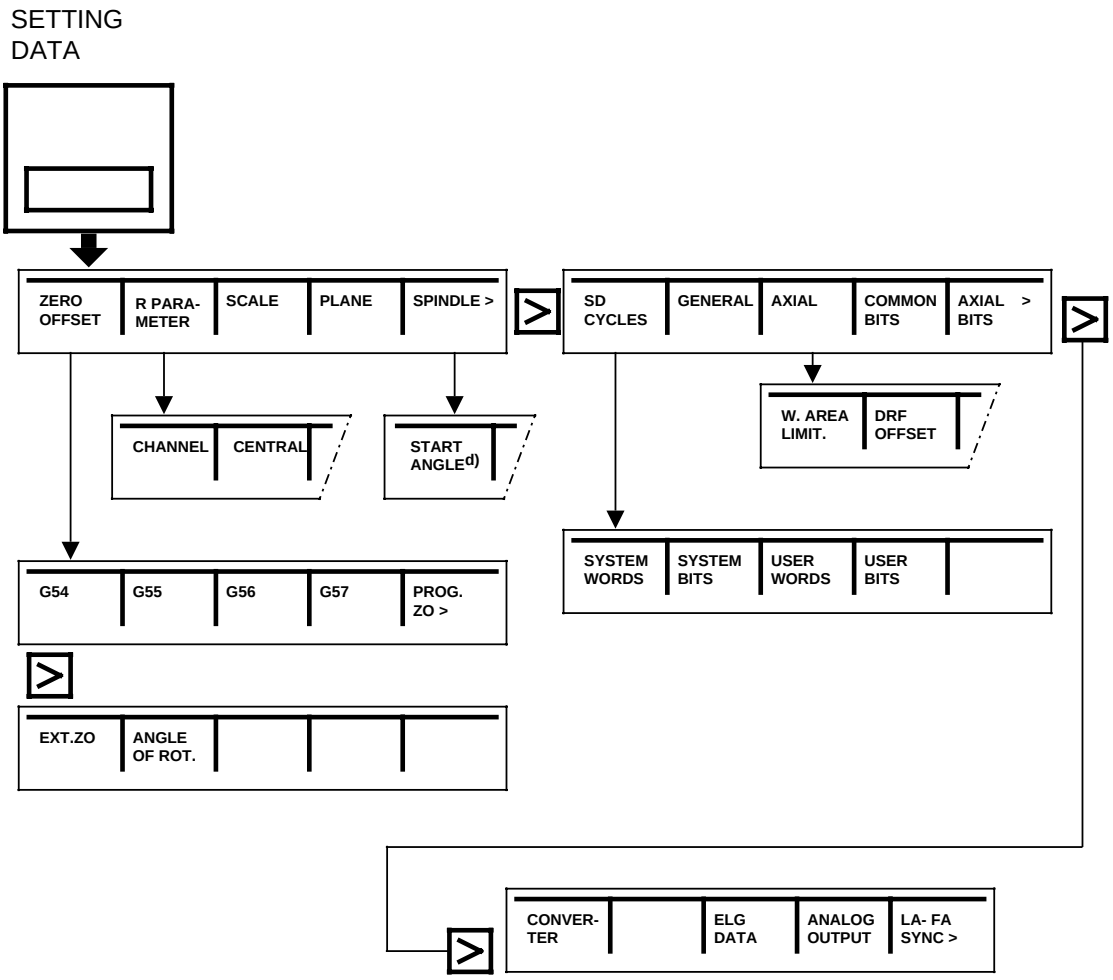
c) up to and including Software Version 6

d) from GA2, Software Version 1

© Siemens AG 1991 All Rights Reserved

2.4.9 Operating functions

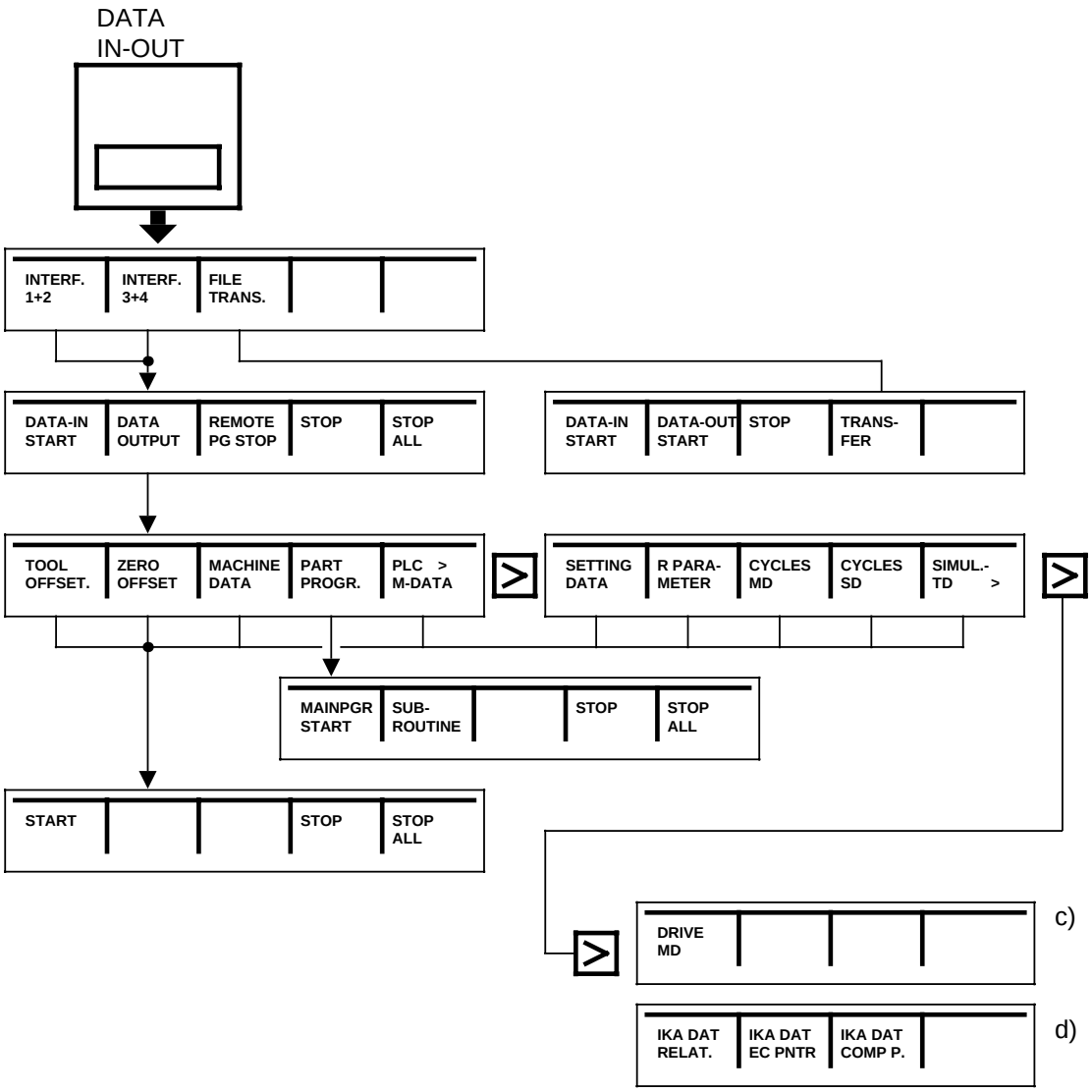
2.4.9.1 SETTING DATA



d) from GA2, Software Version 1

© Siemens AG 1991 All Rights Reserved

2.4.9.2 DATA IN-OUT

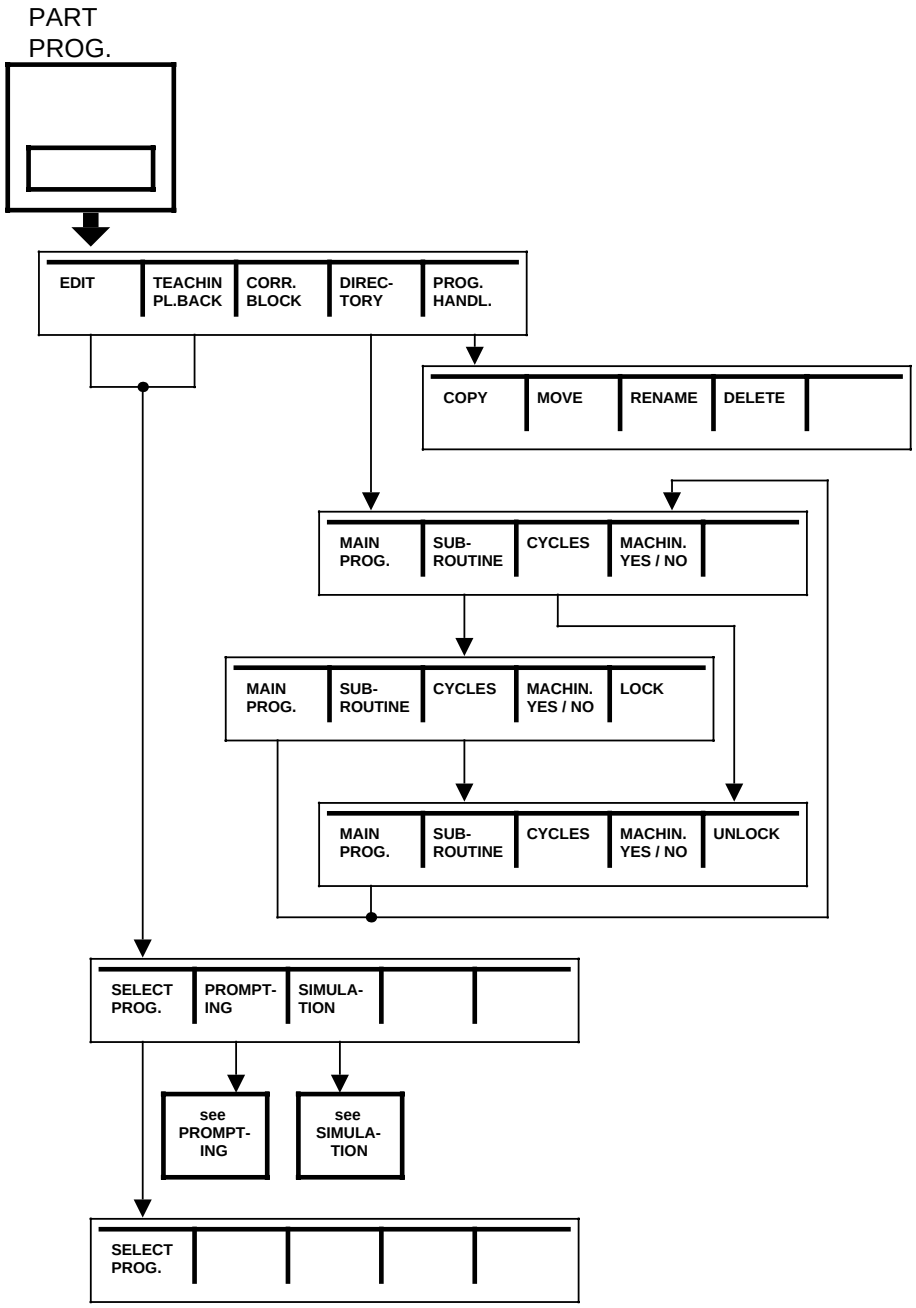


c) up to and including Software Version 6

d) from GA2, Software Version 1

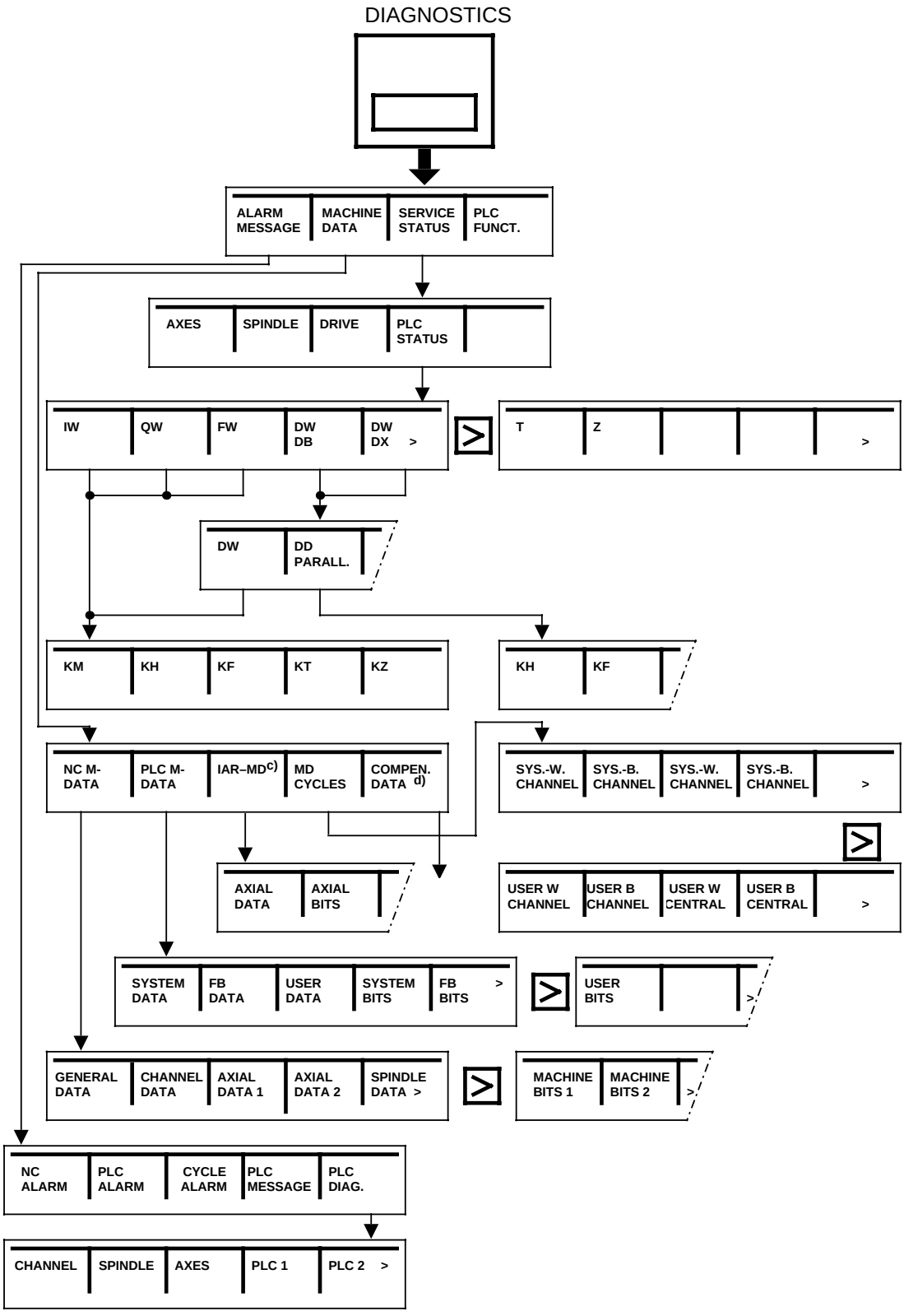
© Siemens AG 1991 All Rights Reserved

2.4.9.3 PART PROGRAM



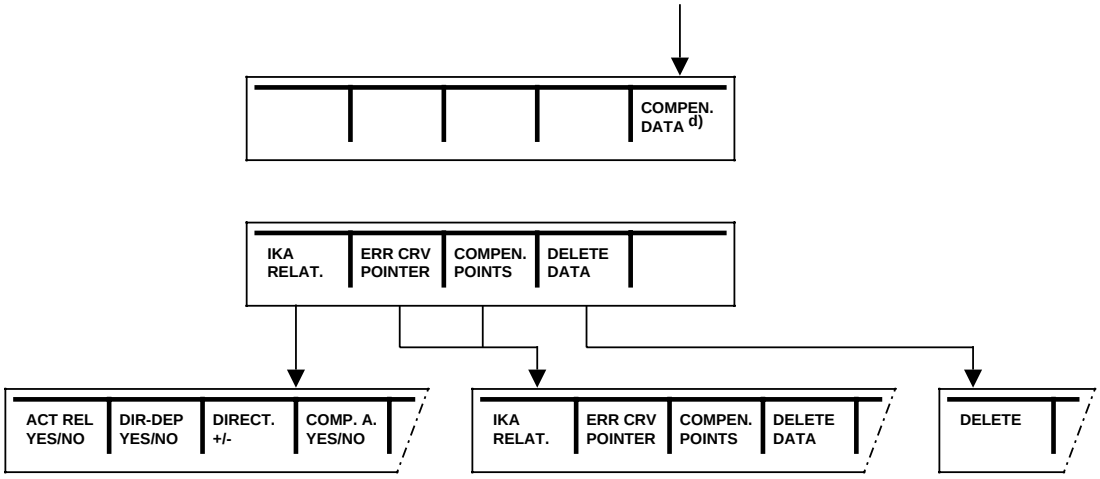
© Siemens AG 1991 All Rights Reserved

2.4.9.4 DIAGNOSTICS



c) up to and including Software Version 6
d) from GA2, Software Version 1

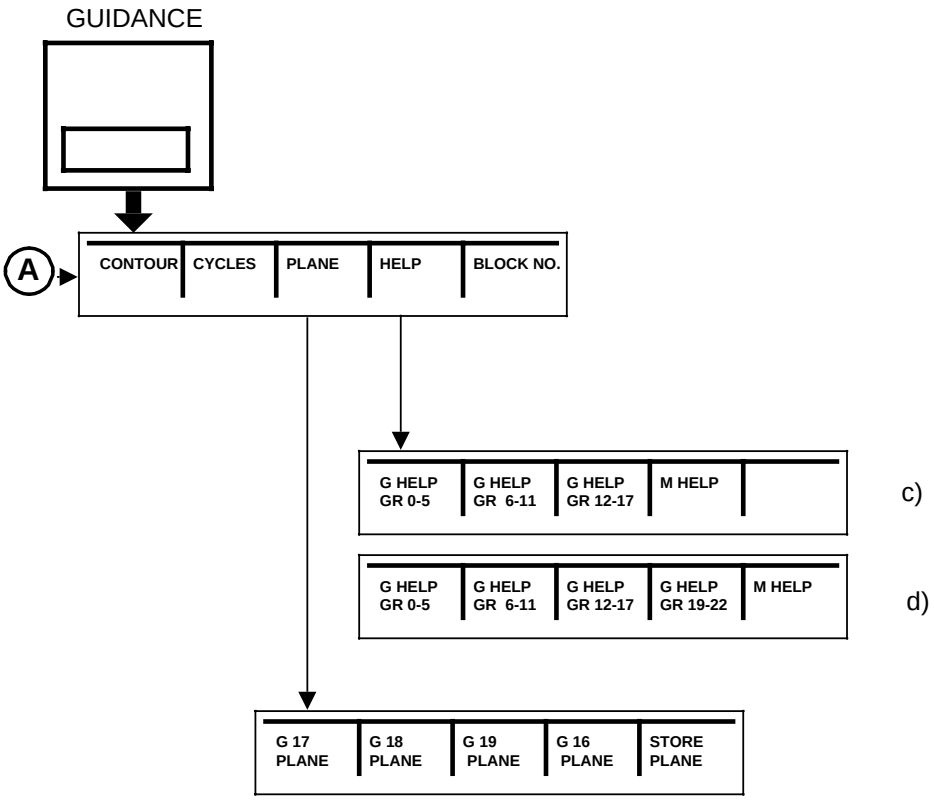
© Siemens AG 1991 All Rights Reserved



d) from GA2, Software Version 1

© Siemens AG 1991 All Rights Reserved

2.4.9.5 GUIDANCE



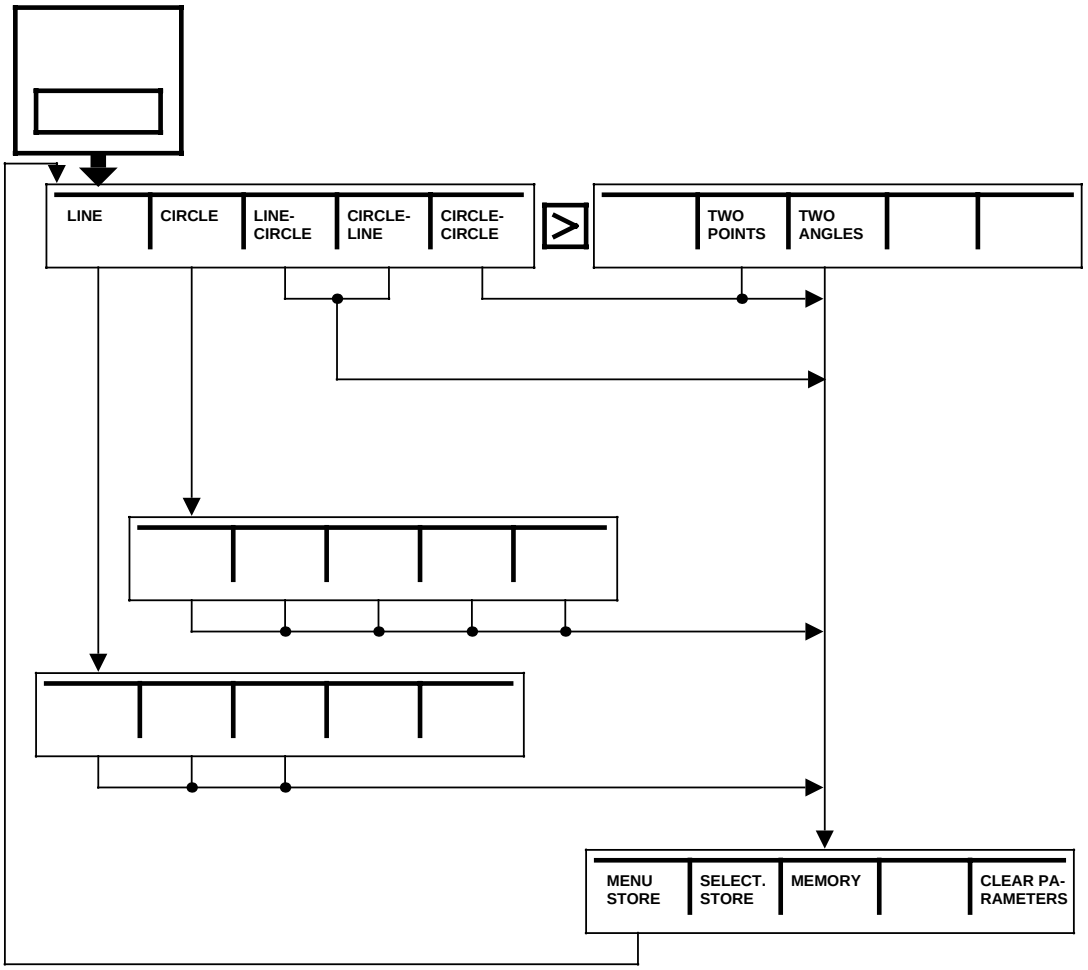
(A) This menu can also be entered by pressing the MENU STORE softkey, see operating function CYCLES softkey.

c) up to and including Software Version 6

d) from GA2, Software Version 1

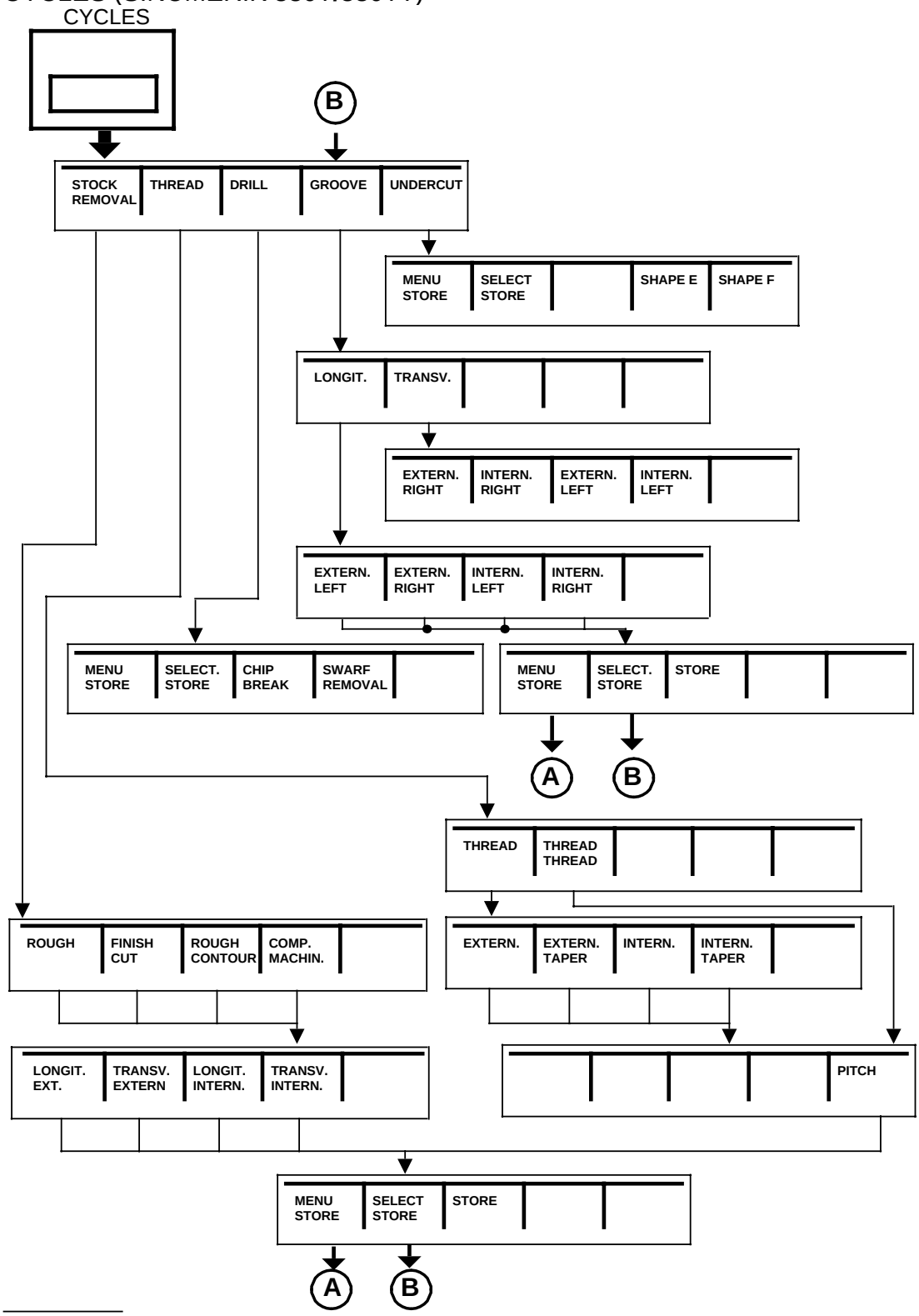
© Siemens AG 1991 All Rights Reserved

CONTOUR



© Siemens AG 1991 All Rights Reserved

CYCLES (SINUMERIK 880T/880TT)

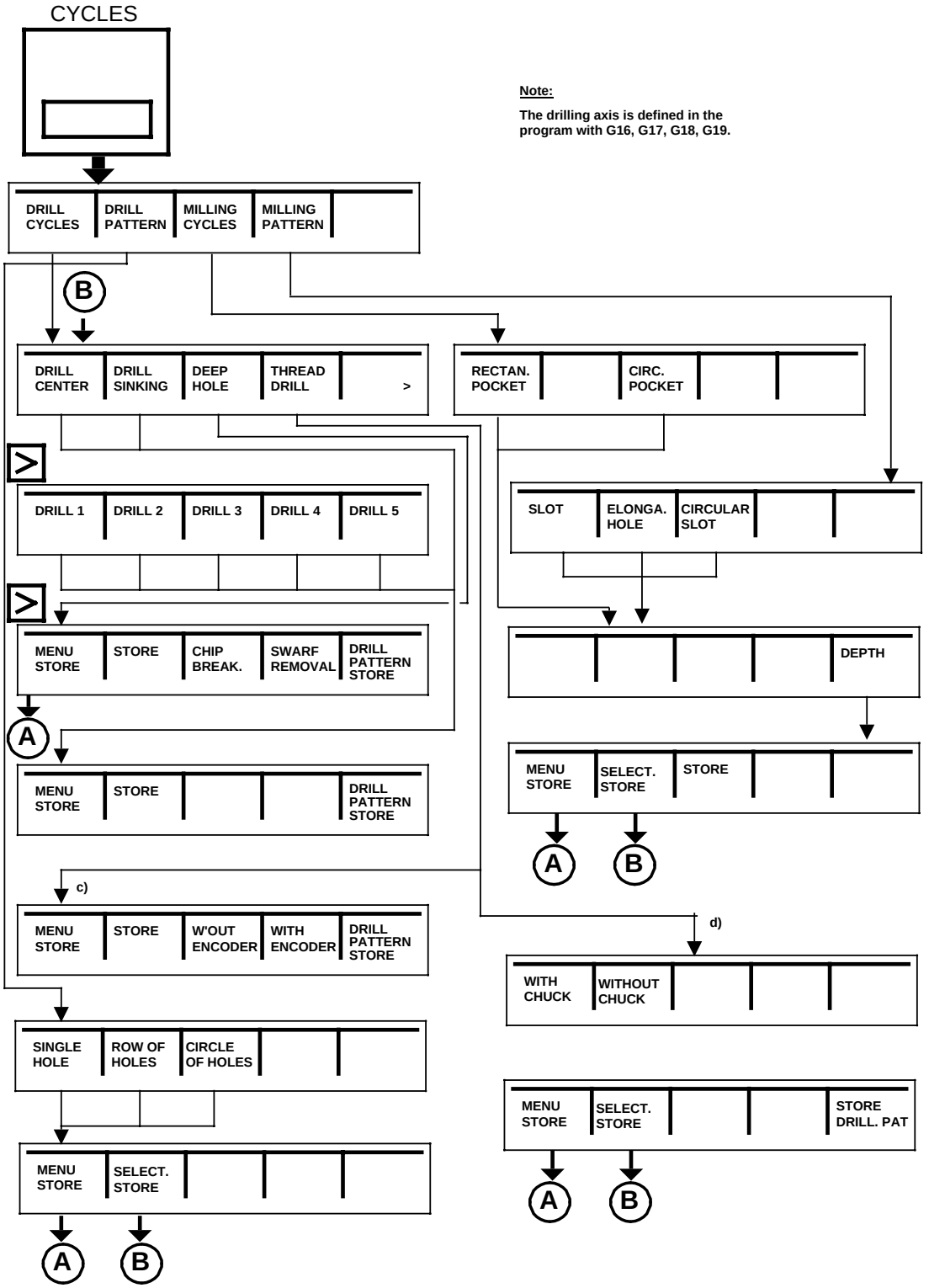


(A) : Menu (A) of the GUIDANCE operating function appears.

(B) : After pressing the SELECT STORE softkey, operation can be continued with the GROOVE softkey.

© Siemens AG 1991 All Rights Reserved

CYCLES (SINUMERIK 880M)



A : Menu A of the GUIDANCE operating function appears.

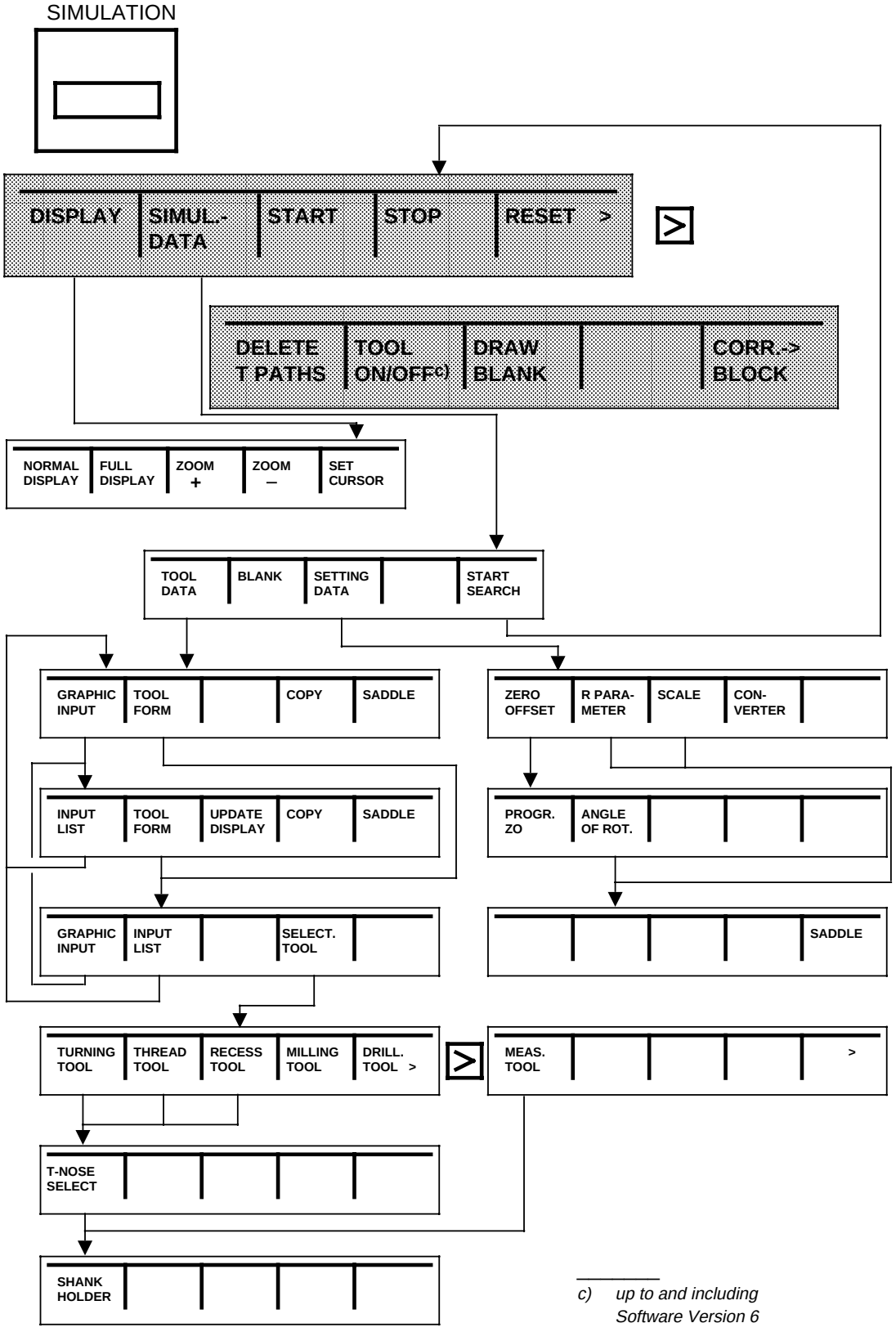
B : After pressing the SELECT STORE softkey operation can be continued with the DRILL CENTRE softkey.

c) up to and including Software Version 6
d) from GA2, Software Version 1

© Siemens AG 1991 All Rights Reserved

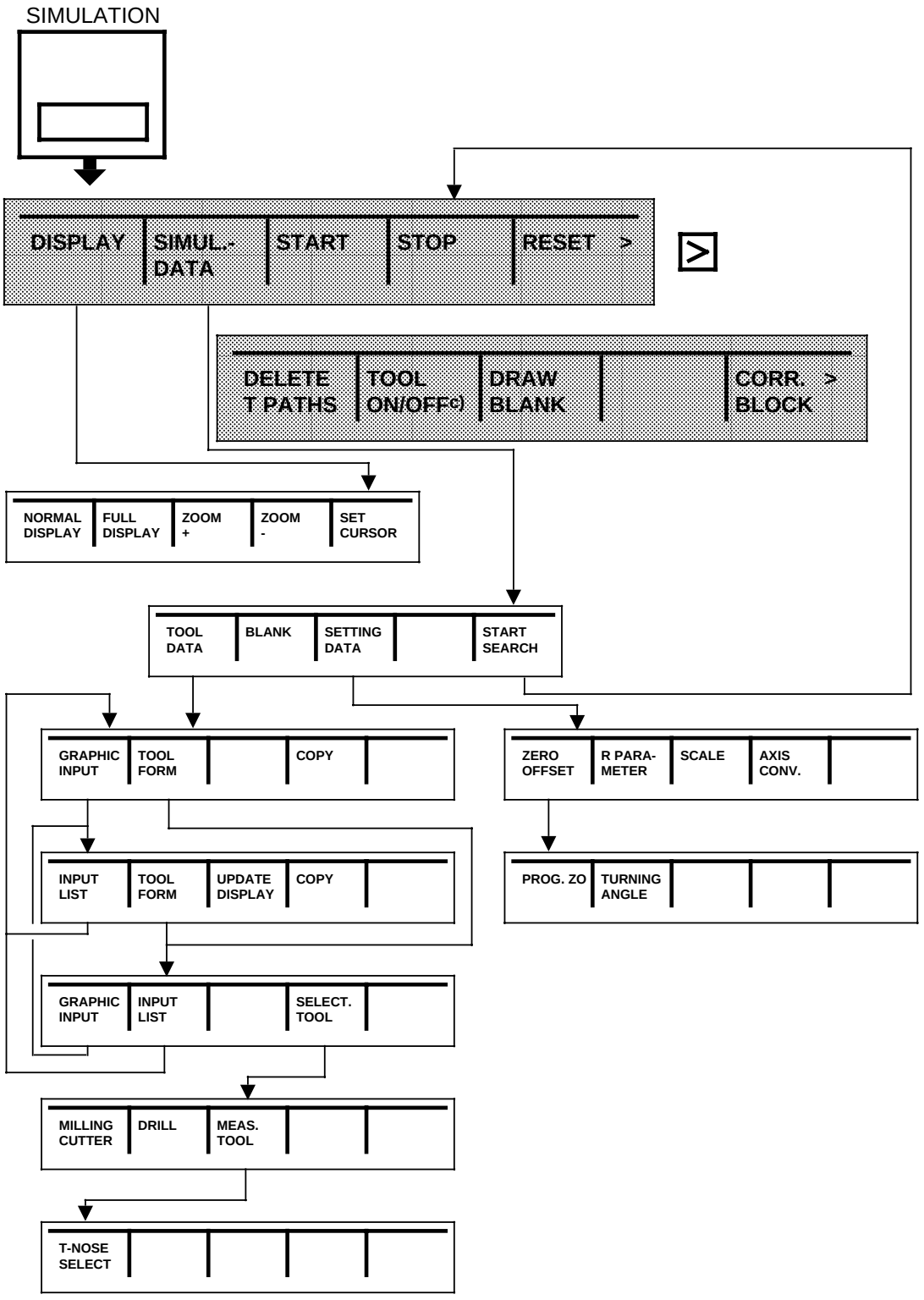
2.4.9.6 SIMULATION

SIMULATION (SINUMERIK 880T, 880TT)



c) up to and including Software Version 6

SIMULATION (SINUMERIK 880M)



c) up to and including Software Version 6

© Siemens AG 1991 All Rights Reserved

2.5 Glossary of softkey functions

The softkey glossary explains in brief the functions that are activated when the softkeys are operated. The softkey functions are listed in alphabetical order.

ALARM MESSAGE

All current alarms are displayed. The alarms for NC and PLC can be selected using further softkeys.

ANALOG OUTPUT

Parameter assignment to analog channels.

ANGLE OF ROTATION

The setting data for the angle of rotation are displayed.

AXIAL

The axial setting data are displayed and can be modified.

AXIAL BITS

The axial setting data bits are displayed and can be modified.

AXIAL DATA

The axial setting data are displayed and can be modified.

BLOCK NUMBER

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

BLOCK SEARCH

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

CLEAR PRE-OFF

(CLEAR PRESET-OFFSET)

You clear the PRESET offset.

COMMON BITS

The common setting data bits are displayed and can be modified.

COMPEN. DATA

Data input for error compensation.
This function is password protected.

COMPEN. POINTS

Transfer of compensation data via external computer:
output of compensation points

CONTOUR

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

- Line
- Line-circle
- Circle
- Circle-line
- Circle-circle
- 2-Point definition
- 2-Angle definition

© Siemens AG 1991 All Rights Reserved

CONVERTER

(CONVERTER OF AXIS/SPINDLE DESIGNATIONS)

When the converter is active the axis/ spindle addresses are converted to other axis/spindle addresses.put of compensation points

COPY

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

CORR.BLOCK

An error in the program is marked with the cursor (correction pointer).

CYCLES

The following are displayed in the appropriate display:

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

Cycles are protected subroutines which are called for frequently used technologies or for machine-specific operations (stock removal cycles, drilling cycles, tool change cycles).

The values required are defined as parameter assignments before the cycles are called (*c*ycle parameter assignment).

DATA IN START

(START DATA INPUT)

You start data input via the universal interface.

DATA IN-OUT

The data (part program, settable zero offsets, machine data, R parameters, tool compensations, PLC program, PLC operator messages, PLC machine data, user programs) are read in from an external device (e.g. punched tape reader) or output to an external device (e.g. printer). Data input and output are handled via interface 1 to 6 as desired. The interface data can be defined separately using setting data.

DATA OUTPUT

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

DEC-SBL YES-NO

(DECODING SINGLE BLOCK ON-OFF)

With "YES", the blocks are processed singly. The function is active at the end of the block in which decoding takes place with the signal present (Program control).

DELETE

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

© Siemens AG 1991 All Rights Reserved

DIRECTORY

The following is displayed:

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

DRF YES-NO

With "YES", the differential resolver function is activated.

DRY RUN YES-NO

With "YES", the axes are traversed at the dry run feedrate, not the programmed feedrate. The dry run feedrate is set via setting data.

EDIT

EDIT leads to SELECT PROGRAM, PROMPTING and SIMULATION.
Editing means:
input of programs into the memory or changing or altering programs already in the memory.

ELG DATA

Speed ratio for the electronic gear (ELG) and for the synchronous spindle.

ERR CRV POINTER

Transfer of compensation data via external computer:
output of error curve pointers

EXTERN. EXEC.

(EXECUTE EXTERNAL PROGRAM)

You can read in and execute part programs from external mass storage via serial interface or via the computer link.

EXT. ZO

(EXTERNAL ZERO OFFSET)

The values supplied by the PLC are displayed.

FILE TRANSFER

(NC FILE TRANSFER)

File transfer with computer link.

GENERAL

The general data are displayed and can be modified.

HELP

Prompting for input of G and M functions is selected.

IKA RELAT.

Transfer of compensation data via external computer:
output of IKA relations

INTERFACE

After DATA IN-OUT, you specify the current interface for data transfer.

LA-FA-SYNCH

Enter synchronisation parameters for leading and following axes.

© Siemens AG 1991 All Rights Reserved

MACH.DATA

(MACHINE DATA)

Using machine data (MD) the control is interfaced to the machine. Certain functions are defined with machine data (e.g. "Dry run feedrate interlocked with a key-operated switch").
A differentiation is made between NC MDs, PLC MDs, IAR MDs and cycle MDs.
Machine data are fixed on installation and cannot be changed by the user.
You select the output of Machine Data with the MACH.DATA softkey after DATA OUTPUT.

MAIN PROGRAM

An overview of the part programs is displayed.

MAINPRG. START

(MAIN PROGRAM START)

You activate the output of part programs via the universal interface.

MEASURE TOOL

Function for measuring tool lengths.

MEASURE WK-PIEC

Function for measuring workpiece lengths.

MOVE

A cycle from the UMS is copied into the part program memory.

NC ALARM

All current NC alarms are displayed (see Section 4).

NC MD

(NC MACHINE DATA)

The NC machine data are displayed.

OVERR. YES-NO

(RAPID TRAVERSE OVERRIDE YES-NO)

With ”Yes”, the override set at the feedrate override switch also becomes effective for rapid traverse.

OVERSTORE

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

PART PROGRAM

The PART PROGRAM softkey leads to:

- EDIT
- TEACH IN/PLAYBACK
- CORRECT BLOCK
- DIRECTORY
- PROGRAM HANDLING

This key is not used to select a program for processing.

After DATA OUTPUT, you can select part program output with the PART PROGRAM softkey.

PLANE

You can select the machining plane with simulation and programming.

PLC ALARM

All current PLC alarms are displayed (see Section 4).

© Siemens AG 1991 All Rights Reserved

PLC MESSAGE

All current PLC messages are displayed (see Section 4).

PLC MD

(PLC MACHINE DATA)

The PLC machine data are displayed. After DATA OUTPUT you select the output of PLC machine data via the PLC MD softkey.

PLC STATUS

The PLC status shows the current state of all inputs, outputs, flags, timers and counters on the CRT:

- IW=Input word
- QW=Output word
- FW=Flag word
- T=Timer
- C=Counter
- KH=Hex constant
- KM=Binary constant

PROCESS YES-NO

Processing of the part program selected is released or locked.

PROGRAM CONTROL

(INFLUENCING THE PROGRAM)

The key leads to the following functions:

- SKIP BLOCK
- DRY RUN FEEDRATE
- PROGRAMMED STOP
- RAPID TRAVERSE OVERRIDE
- DECODING SINGLE BLOCK
- DRF

PROG. ZO

(PROGRAMMABLE ZERO OFFSET)

The values of the programmable zero offset (G58/G59) are displayed on the screen.

PR. HOLD YES-NO

(PROGRAMMED STOP YES-NO)

With ”YES”, processing is stopped at the block in the program which contains the command ”M01”.

PROG. HANDL.

(PROGRAM HANDLING)

You can copy programs in the program memory, transfer them from the UMS into the memory, rename and delete them.You can reorganise the program memory (after deleting programs).

PROMPTING

The operator prompting speeds up and simplifies the input of part programs. Apart from geometric functions (contour) you can also input cycles.

RECUT THREAD

You can recut the thread with a constant pitch.

REMOTE PG STOP

You can close down the link to a remote PG (via the SINEC H1 LAN) and activate the local programming unit.

RENAME

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

R PARAMETERS

You input the R parameters as setting data.

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

© Siemens AG 1991 All Rights Reserved

RESET

Simulation is interrupted and returned to the reset state.

SCALE

Setting data for change of scale are displayed.

SELECT PROGRAM

A part program is selected for coding.

SELECT. STORE

Storing of the values input and jump back to the selection menu.

SELF INSTALL.c)
(SELF-INSTALLATION)

The following is displayed:

Self-installation of the intergrated drive control (IAR) is activated. With this function you can set the control behaviour of the individual axes. The function is protected by a password.

SERVICE STATUS

The associated display shows all the axis and spindle information necessary for service.

SETTING DATA

Using setting data, the operator (user) fixes certain operating states. Setting data are adjustable for:

- Zero offsets
- R parameters
- Scale
- Plane
- Spindle data
- General data (bits)
- Cycle data
- Axial data (bits)
- Converter
- Electronic gear

SIMULATION

To create and test the program the programmed movements are shown on the CRT display. Programming errors are displayed as alarms.

SIMUL. DATA
(SIMULATION DATA)

- Program number
- Type of representation
- Plane
- Simulation control
- NC channel number
- Simulation area
- Block search

SKIP YES-NO
(SKIP BLOCK YES-NO)

With "YES" selected, the blocks marked with an oblique (*/*) are skipped during program processing (Program control).

START

You activate the selected softkey function.

---

c) up to and including Software Version 6

© Siemens AG 1991 All Rights Reserved

START ANGLE

You can change an angle that has already been programmed for cutting a multiple thread.

STOP

You stop the activated softkey function.

STOP ALL

Data transfer is interrupted at all interfaces.

STORE

In "JOG" mode you store the position traversed to (PLAYBACK).

STORE MENU

Storing input values and return to basic menu.

SUBROUT.
(SUBROUTINE)

The following is displayed:

- The stored subroutine numbers
- The number of characters required
- The free memory space

SUBROUTINE START

Output of subroutines via the universal interface is activated.

TEACH IN/PL. BACK
(TEACH IN/PLAYBACK)

With TEACH IN/PLAYBACK you generate a part program block by block. By doing this, you can test the program immediately.

TOOL

You input the tool number T... and the tool compensation number D...

TOOL DATA

Input and monitoring of the simulation tool data.

TOOL OFFSET

The tool offset takes into account the tool dimensions and wear. The tool offsets are stored under a tool offset number, D..., in the tool offset memory.

Via the TOOL OFFSET softkey after DATA OUTPUT, you select the output of tool offsets via the universal interface.

UPDATED BLOCK
(CURRENT BLOCK)

The current block is the block currently being processed. The display shows: The block **before** the current block, the current block and the block **after** the current block (up to 3 lines).

UPDATED VALUES
(CURRENT VALUES)

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

ZERO OFFSET

The settable zero offsets (ZO) are input as setting data. Programmable and external zero offsets are displayed on the CRT (see Section 3.2.5).

© Siemens AG 1991 All Rights Reserved

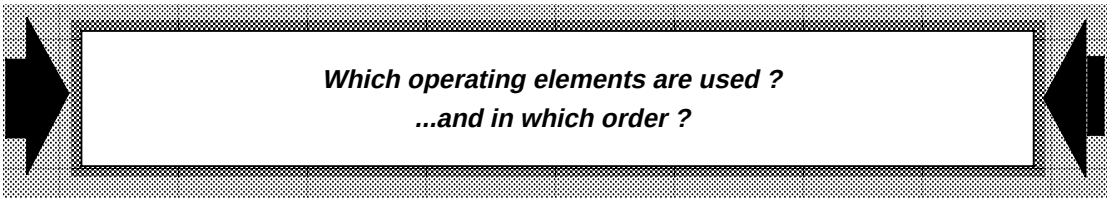
3 Operating Sequences

3.1 Preliminary remarks

In the previous Sections, the functions of the individual operating elements were described in detail.

This Section shows how to use the operating elements in frequently occurring operating sequences.

It explains:



This Section is divided into two parts:

- In the first part (preparation) operating methods are explained that you use before, during or after **program input** or that you use **only once** before you start the control operation.
- In the second part (machining) the operating methods are explained that occur **during the machining process**.

© Siemens AG 1991 All Rights Reserved

3.2 Preparation

3.2.1 Switching on

When you switch the control and machine tool on:

Follow the intructions in the machine tool manufacturer's documentation.

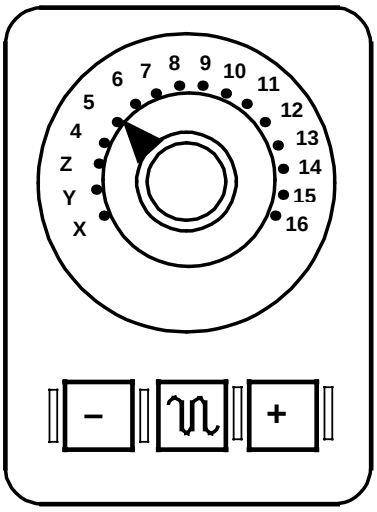
3.2.2 Approach reference point (REFPOINT)

After switching on, approach the reference points for the various axes. This synchronises the control with the machine.

The position of the reference point and the traversing rate are set in machine data by the machine tool manufacturer.

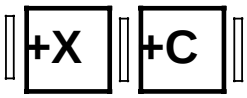
Sequence of operation

Set the operating mode selector switch on the machine control panel to this symbol **REFPOINT**.



With the SINUMERIK 880M, you select the axis to be traversed with the axis selector switch and then press the direction keys. Then you approach the reference point in the individual axes in sequence.

© Siemens AG 1991 All Rights Reserved





The axes are traversed by operating the appropriate direction key(s).





With the "Feed Hold" key you can stop the selected axis before the reference point is reached.

Note:

The selected direction of approach is checked by the control before starting.

→ If you have operated the incorrect direction key (e.g. +X instead of –X), no traverse motion will result as the operation is rejected.

After reaching the reference point the position value referring to the machine zero point is displayed as actual position on the CRT.

The control is now synchronised with the machine.

((Foto aus der Druckschrift SINUMERIK 880, Bedienungsanleitung, Bestellnr. 6ZB5 410-HC01-0BA0 Seite 3–3 einmontieren))

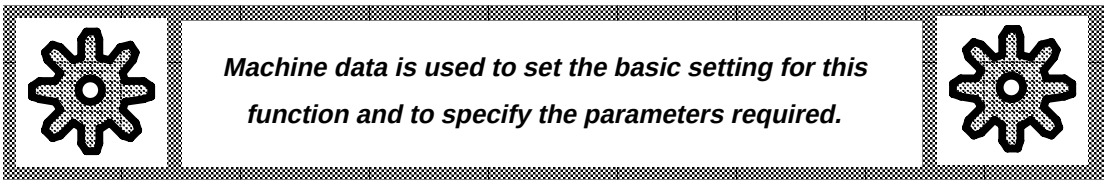
Note:

Using the "Approach reference point for each program" function, it is possible to approach the reference point of the programmed axis through the G function "G74" from a part program. This function can be activated in the **AUTOMATIC** and **MDI-AUTOMATIC** modes.

© Siemens AG 1991 All Rights Reserved

3.2.2.1 Setting-up mode, division-based

The NC axes can be positioned at graduated points using the ”Setting-up mode, division-based” function.



Positioning at a given graduation is possible in **JOG** and **INC FEED 1... 10000** modes.

JOG mode

One of the direction keys ”+” or ”-” is operated. The graduated axis is moved like a normal axis. Once the direction key is released, the axis traverses to the next division position in that direction.

Incremental mode (INC FEED 1... 10000)

One of the ”+” or ”-” direction keys is operated. The graduated axis is traversed incrementally by one division regardless of which INC position (1, 10, 100, 1000 or 10000) has been set at the operating mode switch.

Terms related to the ”Setting-up mode, division-based” function:

Division number:

The division number specifies the number of divisions to be moved along with incremental parts (G91). With absolute parts (G90) it specifies the set division position to be selected.

Number of divisions:

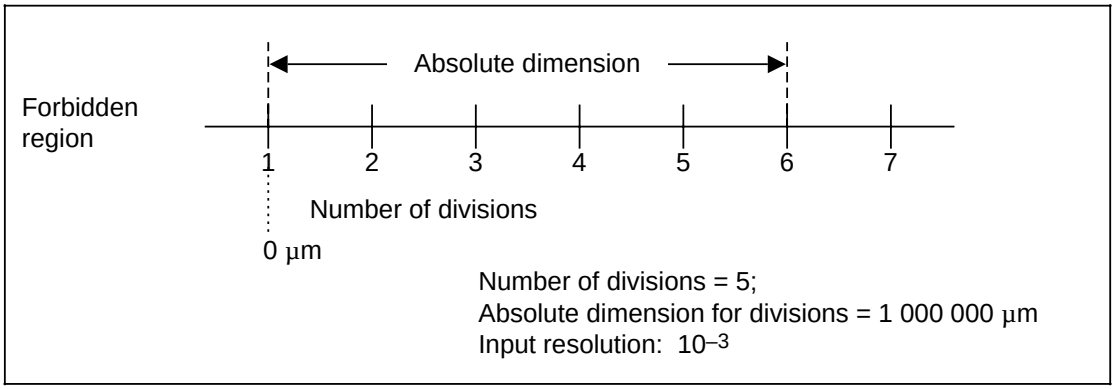
This specifies the number of divisions per each absolute dimension.

Absolute dimension for divisions:

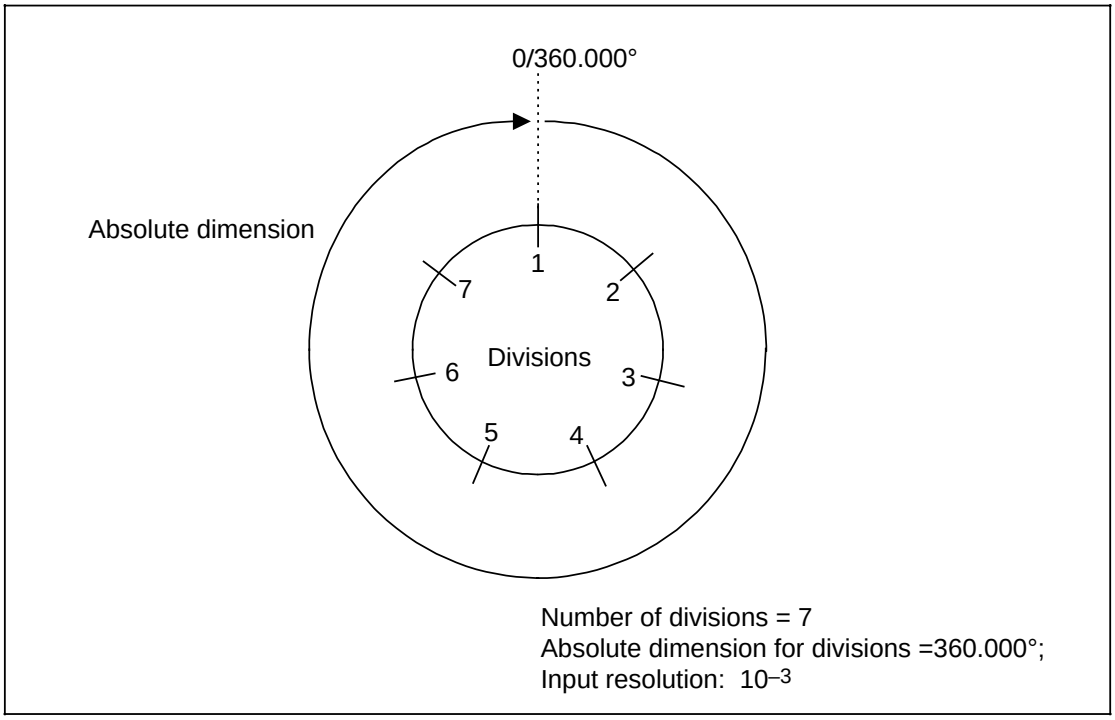
The absolute dimension for divisions specifies the distance to which the number of divisions refers.

© Siemens AG 1991 All Rights Reserved

Number of divisions and absolute dimension for divisions:



Linear axis

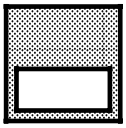


Rotary axis

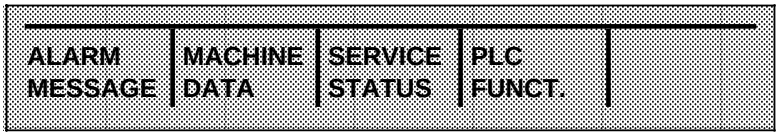
3.2.2.2 Semi-automatic drift compensation

If the  is no longer extinguished after a traversing movement, the drift has exceeded the permissible value and drift compensation *must* be carried out.

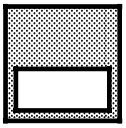
DIAGNOSTICS



Operate the DIAGNOSTICS softkey.

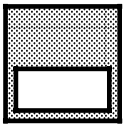


MACH. DATA



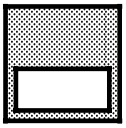
Operate the MACHINE DATA softkey.

NC MD



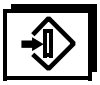
Operate the NC MD softkey.

AXIAL DATA



Operate the AXIAL DATA softkey

272*



Use this key to call machine data 272*.
Set the cursor to the desired axis.

* = 0 1st axis
* = 1 2nd axis
* = 2 3rd axis
* = 3 4th axis etc.



Operate the "Modify word" key.

The new compensation value is displayed as machine data.

© Siemens AG 1991 All Rights Reserved

3.2.3 Tools

The Machine Tool Manufacturer's Documentation should be consulted about tooling.

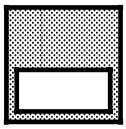
3.2.4 Tool offset

3.2.4.1 Input of tool offset

Sequence of operation

In any operating mode:

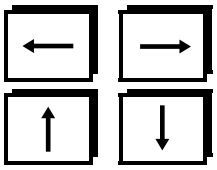
TOOL OFFSET



((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-HC01-0BA0
Seite 3–7 einmontieren))

This input screen form will help you to input tool offset data. For this, the following keys can be used in addition to the numeric keys:





With these keys you position the cursor to any input field.

Each value input is stored by pressing the yellow input key and then appears on the CRT in the respective input field.

© Siemens AG 1991 All Rights Reserved



With the key "Modify word", you can change the current value by adding or subtracting.

Example:

Actual value: 50

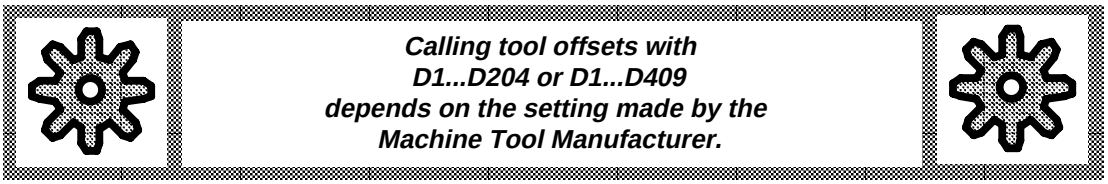


New value: 45

Explanation of the input screen form

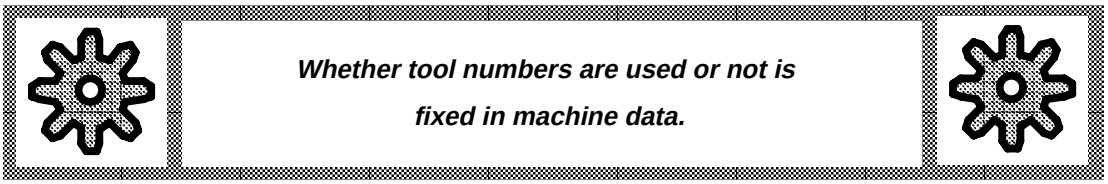
D 2

Name of the tool offset memory ("D 2" in this example).
SINUMERIK 880 has 204 (409) offset memories.
You can therefore address the tool offset numbers D1 to D204 (D409), and store offset data under each one.

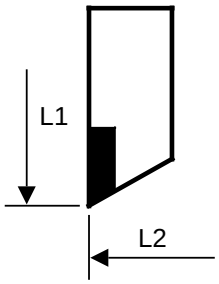
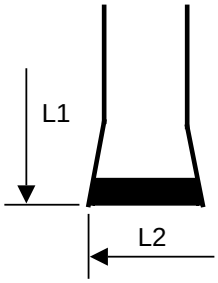
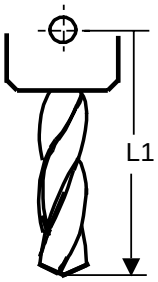
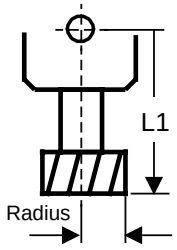
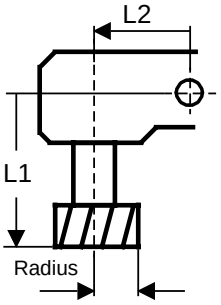


Tool parameter 0:
Tool number

The input field marked "0" is intended for the input of a tool number of up to 8 digits.
Normally entries are not required except when using flexible tool management.

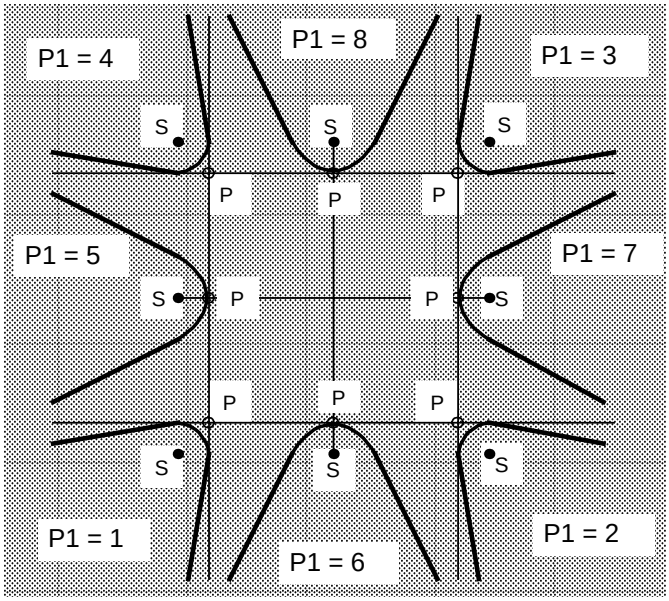


© Siemens AG 1991 All Rights Reserved

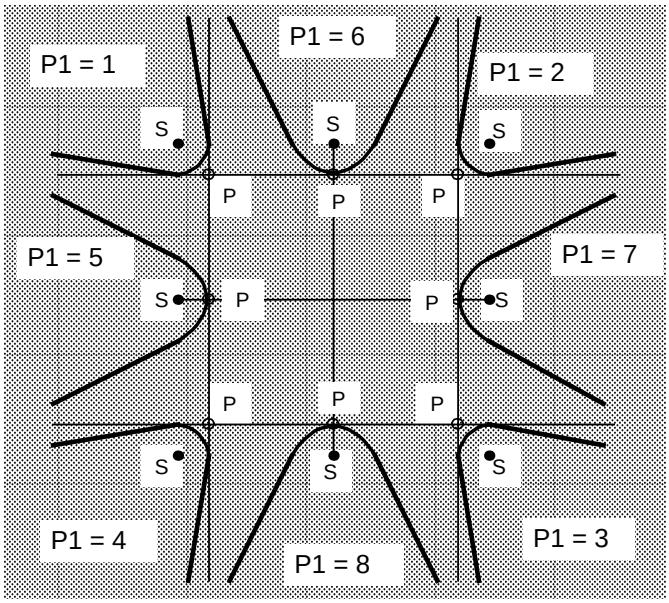
Tool parameter 1: Tool type	**Tool**		**Tool type P1**
	1. Lathe tool Example: **Turning tool, facing tool**		1 ... 9
	2. Lathe tool Example: **Grooving tool** (tool nose left)		1 ... 9
	3. Tools with length compensation only Example: **Drill**		10
	4. Tools with radius compensation and one length compensation Example: **End mill**		20
	5. Tools with radius compensation and two length compensations Example: **Angular milling cutter**		30

© Siemens AG 1991 All Rights Reserved

With SINUMERIK 880T (lathe tools) the tool nose centre point "S" (Reference Point: "P") is entered in the "Tool type" input field under the identification "1". The input values "P1 = 1" to "P1 = 9" are available:



This diagram shows you which values you should select for **machining behind the turning centre.**
P1=9 is valid when the tool nose centre point "S" is the reference point.



This diagram shows which values you should select for **machining in front of the turning centre.**
P1=9 is valid when the tool nose centre point "S" is the reference point.

Note:

Type 40 (milling cutter with 5D tool length correction) is optionally available in addition to the five tool types

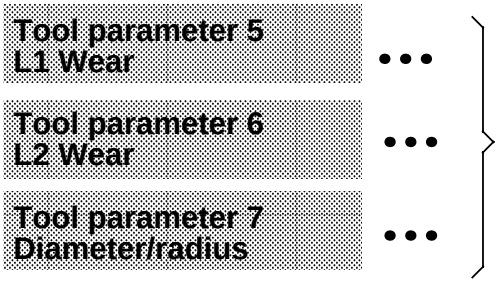
Tool parameter 2
L1 Geometry ••••

Tool parameter 3
L2 Geometry ••••

Tool parameter 4
Diameter/radius ••••

Under tool parameters "**2**" and "**3**", you enter the geometry values of the tool in the input field.
Under tool parameter "**4**" you enter the value of the (tool nose) radius.

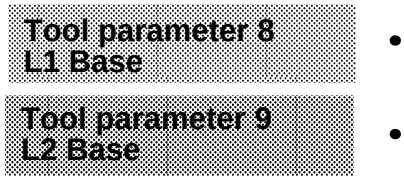
© Siemens AG 1991 All Rights Reserved



Under tool parameters "**5**" to "**7**", you can enter the wear data of the tool in the input screen form (not obligatory).

Minimum and maximum input values can be modified using the softkeys ”NC MD, COMMON DATA, Nos. 208 ... 209”

The input of the geometry and wear data can be inhibited with a key-operated switch. Please see the Machine Tool Manufacturer's Documentation!



Tool parameters **"8"** and **"9"** are reserved for special uses. The "Base dimension" permits an additional tool length offset.

The standard distribution of the identifiers (0 to 9) can be raised to a maximum of 16 using machine data. Please see the Machine Tool Manufacturer's Documentation.

3.2.4.2 Deleting/modifying an individual offset value

You select the desired tool offset number under which you intend to delete***/***modify an offset value and position the cursor at the input field to be deleted***/***modified (see also Section 3.2.4.1). Using the numerical keys you enter the new value or the value "zero".

© Siemens AG 1991 All Rights Reserved

3.2.4.3 Deleting all offset values of a tool offset number D...

a) You would like, for example, to delete the values stored under tool offset number "D5".

Sequence of operation	
5	Key in "5".
	You operate the "Delete word**/**block" key.

If you select the tool offset number "D5", using either the "Page" key , or by entering the number "5" and operating the search key  you will find that all the values in this offset number have been set to ”zero”.

b) You would like, for example, to delete the values stored under tool offset numbers ”D4” to ”D9”.

Sequence of operation	
4 = 9	Key in "4=9".
	Operate the "Delete word**/**block key".

On calling the tool offset numbers "D4" to "D9", as described in a) above, you will find that all offset values have been set to ”zero”.

© Siemens AG 1991 All Rights Reserved

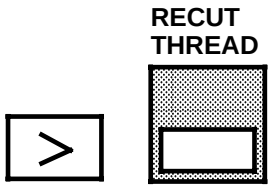
3.2.4.4 Recut thread

The function EXTENDED THREAD PACKAGE (Recut thread) is an option.

With this function you can reclamp a precut thread and recut it.
Only threads with a **constant** pitch can be recut.

You are in one of the following modes:

JOG/INC FEED 1 ... 10 000

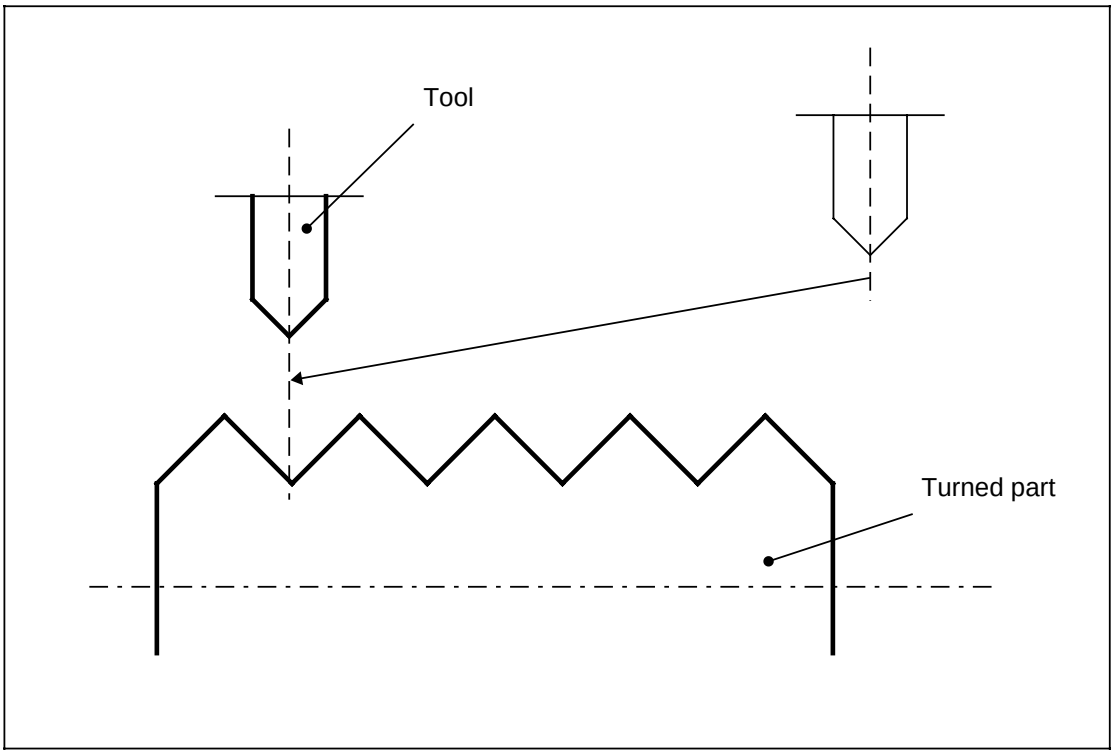


((Mitgeliefertes Foto
Nr. 3–13 einmontieren))

You must take the following steps to recut a thread:

1. Clamp the workpiece into position.
2. Position the tool manually at the thread in such a way that the tool point is positioned directly above the root of the thread.
3. Press softkey "STORE POSIT.".
   This sets the "offset angle" and the "start angle (G92A)" to zero.
4. Start NC program (AUTOMATIC or MDA).
   The offset angle is calculated in the first thread cutting block and compensated for.

© Siemens AG 1991 All Rights Reserved



Thread recutting

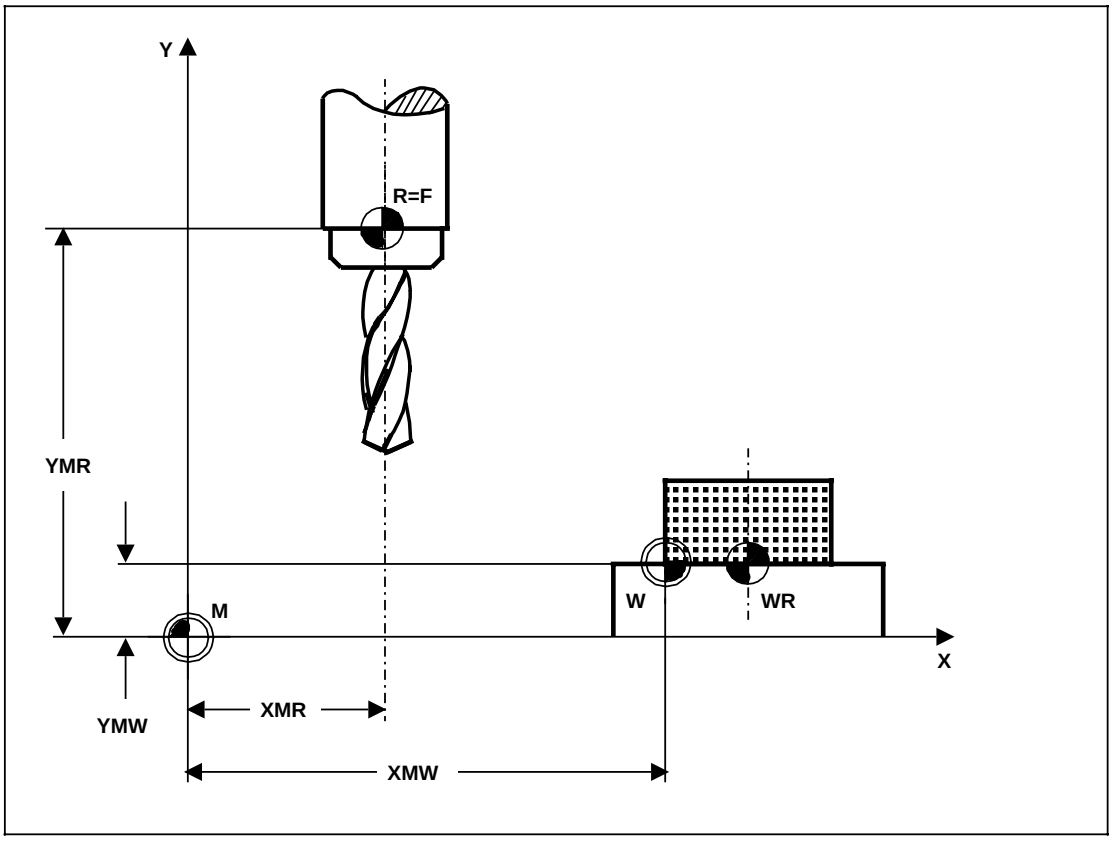
3.2.5 Zero offset

The actual value memory, and therefore the actual value display, are referred to the machine zero point **"M"** after approaching the reference point.

The machining program for the workpiece is referred to the workpiece zero point "**W**".

Machine zero point "**M**" and workpiece zero point "**W**" are not identical. The dimensions between the machine zero point "**M**" and the workpiece zero point "**W**" can vary, depending upon the type and fixing of the workpiece. The zero offset will allow for this during program operation.

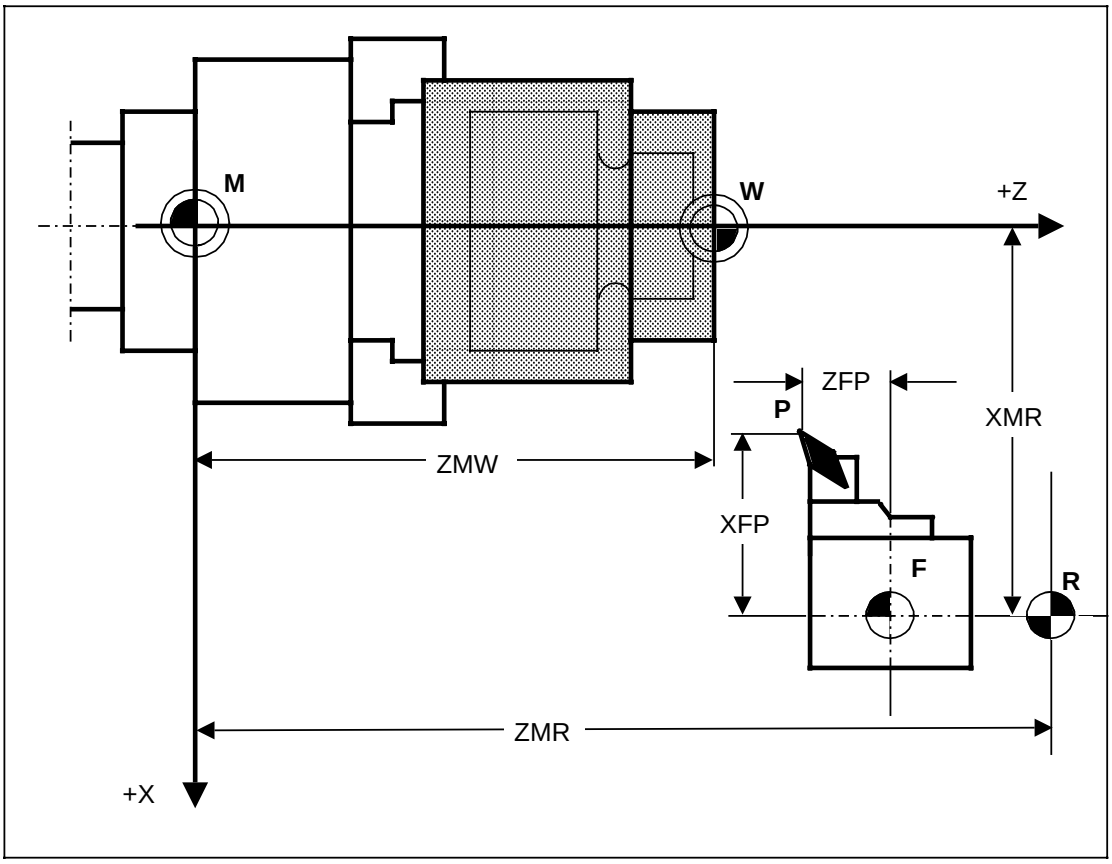
© Siemens AG 1991 All Rights Reserved



Zero offset with SINUMERIK 880M

M	Machine zero point
W	Workpiece zero point
R	Machine reference point
F	Saddle reference point
WR	Workpiece reference point
XMR, YMR	Reference point coordinates
XMW, YMW	Zero offset

© Siemens AG 1991 All Rights Reserved



Zero offset with SINUMERIK 880T

P	Tool setting point
M	Machine zero point
W	Workpiece zero point
R	Machine reference point
F	Saddle reference point
XMR, ZMR	Reference point coordinates
ZMW	Zero offset
XFP	Tool geometry L1
ZFP	Tool geometry L2

© Siemens AG 1991 All Rights Reserved

3.2.5.1 Settable zero offsets

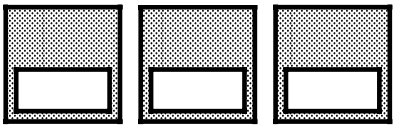
Input the values for the settable zero offset into the control as "SETTING DATA". You can select 4 settable zero offsets for each axis (”G54” to ”G57”).

The zero offsets entered are activated in the part program called.

Sequence of operation

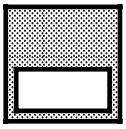
In any operating mode:

SETTING DATA **ZERO OFFSET** **G54**



((Mitgeliefertes Foto
Nr. 3–17 einmontieren))

G . .



Using a softkey you can change to another zero offset (G55 ... G57).

In the input screen form displayed you now enter the offset value for each axis. You can mark any input field with the cursor keys.



You return to the next softkey menu up.

© Siemens AG 1991 All Rights Reserved

3.2.5.2 Programmable zero offset

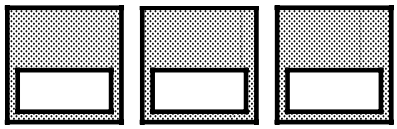
The values for the G58 and G59 offsets are stored in the program. You will find information on the programming of these zero offsets in the "Programming Guide" for SINUMERIK 880.

The programmed zero offset for the current part program % ... can be displayed on the CRT.

Sequence of operation

You have selected the **AUTOMATIC** operating mode and have called a program %...

SETTING DATA **ZERO OFFSET** **PROG.ZO**



((Mitgeliefertes Foto
Nr. 3–18 einmontieren))



You return to the next softkey menu up.

© Siemens AG 1991 All Rights Reserved

3.2.5.3 External zero offset

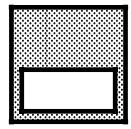
The "External zero offsets" are values transferred from the PLC. These values can also be displayed on the screen.

Sequence of operation

You have selected the **AUTOMATIC** operating mode and have called a program %...

SETTING DATA **ZERO OFFSET** >

EXT. ZO





((Mitgeliefertes Foto
Nr. 3–19 einmontieren))

You return to the next softkey menu up.

© Siemens AG 1991 All Rights Reserved

3.2.6 Electronic gear (ELG)



The "ELECTRONIC GEAR" function is an Option.

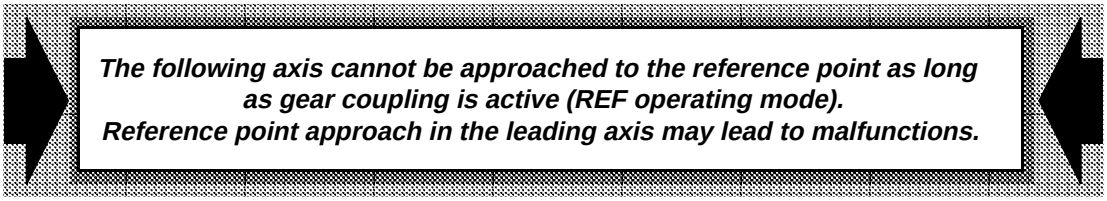
The ELECTRONIC GEAR function allows precise coupling of axes during machining and enables variable transmission ratios to be set.

The gear coupling consists of leading axes (LA) and following axes (FA). The travel commands and motions of the leading axes are transferred to the following axes.

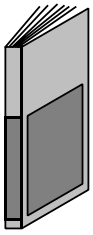
Leading axes may be real or fictitious axes:
Real axes actually perform the motions, fictitious axes do not.

The linear relationship between a leading axis and the relevant following axis is defined by the transmission ratio $K_{ü}$.

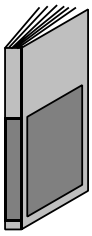
Up to 3 following axes can be defined for the SINUMERIK 880. Each following axis can be simultaneously influenced by 3 real and 2 fictitious leading axes.



The following axis cannot be approached to the reference point as long as gear coupling is active (REF operating mode).
Reference point approach in the leading axis may lead to malfunctions.



Please refer to the "Electronic Gear (ELG)" Function Manual for more detailed information on the functions described in this Operator's Guide.



Note:

All functions described in this Operator's Guide can also be activated and deactivated by the PLC user program via the interface.

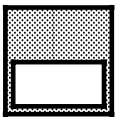
© Siemens AG 1991 All Rights Reserved

Input transmission parameters

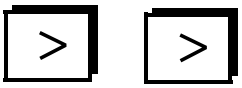
Transmission parameters are used to determine the transmission ratio $K_ü$. The motion a following axis performs with regard to the leading axis/axes is calculated on the basis of this transmission ratio. If a following axis is influenced by several leading axes, a separate transmission ratio $K_ü$ is taken into account for each leading axis.

Sequence of operation

SETTING DATA



Operate the SETTING DATA softkey in any operating mode.



Extend the displayed softkey menu twice using this key.

ELG DATA



You then operate the ELG DATA softkey.

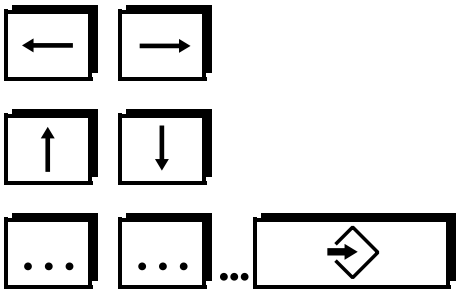
The input screen form showing the transmission parameters is displayed.

((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-HC01-0BA0
Seite 3–23 einmontieren))

© Siemens AG 1991 All Rights Reserved

You enter the desired values for numerator and denominator of the transmission ratio in this input screen form (see Table ”Transmission parameters”).

Proceed as follows:



You shift the input field to the desired position using the cursor keys.

You enter the relevant values using the numeric keys.

Each value is transferred using the input key.

Note:

- Maximum input value 99 999 999 (8 digits)
  Max. 7 places behind the decimal point
  (e. g. 0.1234567)
- Maximum $K_{ü}$ 1000
- For the input transmission parameters to be taken into account for machining, they must first be calculated and then activated.

Setting data no.			Transmission parameters	LA	$K_{ü}$
FA1	FA2	FA3			
SD10	SD20	SD30	Numerator	1	1st fictitious leading axis
SD11	SD21	SD31	Denominator		
SD12	SD22	SD32	Numerator	2	2nd fictitious leading axis
SD13	SD23	SD33	Denominator		
SD14	SD24	SD34	Numerator	3	1st real leading axis
SD15	SD25	SD35	Denominator		
SD16	SD26	SD36	Numerator	4	2nd real leading axis
SD17	SD27	SD37	Denominator		
SD18	SD28	SD38	Numerator	5	3rd real leading axis
SD19	SD29	SD39	Denominator		

Table showing the transmission parameters

© Siemens AG 1991 All Rights Reserved

Calculate transmission parameters

Sequence of operation

You call the input screen form ”Common bits setting data” using the COMMON BITS softkey.

(The sequence of operation is described in Section 3.2.14.1 ”Setting data bits”.)

In this input screen form you select the SD number for the relevant following axis

- 5004 FA1
- 5006 FA2
- 5008 FA3

and set

- bit 7 to 1 Calculate new $K_{ü}$

Activate transmission parameters

Sequence of operation

You call the input screen form ”Common bits setting data” using the COMMON BITS softkey.

(The sequence of operation is described in Section 3.2.14.1 ”Setting data bits”.)

In this input screen form you select the SD number for the relevant following axis

- 5005 FA1
- 5007 FA2
- 5009 FA3

and set

- bit 7 to 1 Activate new $K_{ü}$

© Siemens AG 1991 All Rights Reserved

Switch gear coupling on/off

The following operations must be properly performed before gear coupling can be switched on:

- Input of transmission parameters
- Calculation of new transmission ratios
- Activation of new transmission ratios

Sequence of operation

You call the input screen form ”Common bits setting data” using the COMMON BITS softkey.

(The sequence of operation is described in Section 3.2.14.1 ”Setting data bits”.)

In this input screen form you select the SD number for the relevant following axis

- 5004 FA1
- 5006 FA2
- 5008 FA3

and set

- bit 0 to 1 Switch on gear coupling

Note:

- It is normal for the gear coupling to be switched on while the axes are stationary.

To switch off gear coupling you set

- bit 0 to 0 Switch off gear coupling

Note:

The following axis traverses to the synchronisation position specified in the synchronisation parameters, if synchronisation is enabled when gear coupling is switched off.

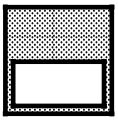
© Siemens AG 1991 All Rights Reserved

Input synchronisation parameters

Synchronisation enables relative positions to be established between the following axis and one programmable leading axis.

Sequence of operation

SETTING DATA



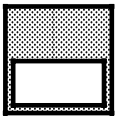
You operate the SETTING DATA softkey in any operating mode.





You extend the displayed softkey menu twice using this key.

LA–FA SYNC



You then operate the SYNCHRONISATION PARAMETERS LEADING AXIS-FOLLOWING AXIS softkey.

The input screen form showing the synchronisation data is displayed.

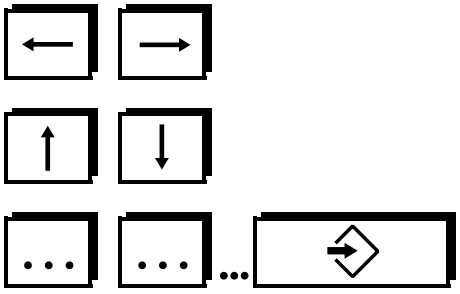
((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-HC01-0BA0
Seite 3–28 einmontieren))

© Siemens AG 1991 All Rights Reserved

You enter the desired values in this input screen form (see Table ”Parameters for synchronising the following axis with the leading axis”):

- Synchronisation position of following axis
- Synchronisation position of leading axis
- Number of leading axis
  - 0 The position of the leading axis is not taken into account
  - 1 Following axis synchronised with the 1st fictitious leading axis
  - 2 Following axis synchronised with the 2nd fictitious leading axis
  - 3 Following axis synchronised with 1st real leading axis
  - 4 Following axis synchronised with 2nd real leading axis
  - 5 Following axis synchronised with 3rd real leading axis

You proceed as follows:



You shift the input field to the desired position using the cursor keys.

You enter the relevant values using the numeric keys.

You transfer each value using the input key.

Note:

For the entered synchronisation parameters to be taken into account for machining, synchronisation must first be enabled.

SD no.	Designation
56	Synchronisation position FA1
57	Synchronisation position of LA for FA1
58	LA number for synchronisation with FA1
59	Synchronisation position FA2
60	Synchronisation position of LA for FA2
61	LA number for synchronisation with FA2
62	Synchronisation position FA3
63	Synchronisation position of LA for FA3
64	LA number for synchronisation with FA3

Table Parameters for synchronising following axis with leading axis

© Siemens AG 1991 All Rights Reserved

Enable/disable synchronisation

Sequence of operation

You call the input screen form ”Common bits setting data” using the COMMON BITS softkey.

(The sequence of operation is described in Section 3.2.14.1 ”Setting data bits”.)

In this input screen form you select the SD number for the relevant following axis

- 5004 FA1
- 5006 FA2
- 5008 FA3

and set

- bit 1 to 1 Enable synchronisation

The synchronisation between leading and following axis is now enabled and can be started by the following control commands:

- ”COP ON” (Switch on gear coupling)
- ”COP OFF” (Switch off gear coupling)
- ”Activate new $K_ü$” (Activate transmission ratios; when gear coupling is activated)
- ”Synchronisation START” (Start synchronisation; when gear coupling is activated)

If synchronisation is not to be enabled you set

- bit 1 to 0 Synchronisation not enabled

© Siemens AG 1991 All Rights Reserved

Start synchronisation

The following preconditions must be met before synchronisation can be started:

- FA overlay must have been enabled (see next function ”Switch on**/**off FA overlay”).
- A free NC channel must be available in the mode group for the following axis to be able to approach the synchronisation position.

Sequence of operation

You call the input screen form ”Common bits setting data” using the COMMON BITS softkey.

(The sequence of operation is described in Section 3.2.14.1 ”Setting data bits”.)

In this input screen form you select the SD number for the relevant following axis

- 5005 FA1
- 5007 FA2
- 5009 FA3

and set

- bit 5 to 1 Start of synchronisation

Leading axes not involved in synchronisation must be stationary.

© Siemens AG 1991 All Rights Reserved

Switching FA overlay on/off

Setpoint suppression for the following axis is activated when the ”FA overlay” function is switched on. This enables the following axis to perform additional traversing motions apart from those predetermined by the leading axes.

Sequence of operation

You call the input screen form ”Common bits setting data” using the COMMON BITS softkey.

(The sequence of operation is described in Section 3.2.14.1 ”Setting data bits”.)

In this input screen form you select the SD number for the relevant following axis

- 5004 FA1
- 5006 FA2
- 5008 FA3

and set

- bit 2 to 1 FA overlay ON

To switch off FA overlay you set

- bit 2 to 0 FA overlay OFF

Note:

FA overlay should only be used if the following axis is to be traversed incrementally with active coupling (incremental dimension, G91). With absolute positioning (absolute dimension, G90), the following axis would move to a wrong position.

© Siemens AG 1991 All Rights Reserved

Enable/disable emergency retraction

With coupled axes, any faults in coupled motions may result in damage to the workpiece, tool or the machine itself. Such damage can be largely avoided or minimized if the coupled axes can be very quickly separated when failure occurs using suitable precautions (emergency retraction).

Sequence of operation

You call the input screen form ”Common bits setting data” using the COMMON BITS softkey.

(The sequence of operation is described in Section 3.2.14.1 ”Setting data bits”.)

In this input screen form you select the SD number for the relevant following axis

- 5004 FA1
- 5006 FA2
- 5008 FA3

and set

- bit 5 to 1 Enable emergency retraction

If emergency retraction is not to be enabled you set

- 5 to 0 No enable of emergency retraction

Note:

The ”Enable emergency retraction” function should be switched off (set bit 5 to 0) while changeover is effected to a new transmission ratio.

© Siemens AG 1991 All Rights Reserved

Acceleration limitation synchronous/not synchronous

After an acceleration process, if the "Acceleration limitation synchronous" function is active, the following axis will follow the motions of the leading axes in position synchronism again, even if synchronism could not be maintained during acceleration.

Sequence of operation

You call the input screen form ”Common bits setting data” using the COMMON BITS softkey.

(The sequence of operation is described in Section 3.2.14.1 ”Setting data bits”.)

In this input screen form you select the SD number for the relevant following axis

- 5004 FA1
- 5006 FA2
- 5008 FA3

and set

- bit 6 to 1 Acceleration limitation synchronous

If the ”Acceleration limitation synchronous” function is not to be activated you set

- bit 6 to 0 Acceleration limitation not synchronous

© Siemens AG 1991 All Rights Reserved

Switch the balancing controller on/off

The balancing controller should be switched on when gear coupling is activated to ensure perfect synchronism in the event of malfunctions.

Sequence of operation

You call the input screen form ”Common bits setting data” using the COMMON BITS softkey.

(The sequence of operation is described in Section 3.2.14.1 ”Setting data bits”.)

In this input screen form you select the SD number for the relevant following axis

- 5004 FA1
- 5006 FA2
- 5008 FA3

and set

- bit 4 to 1 Balancing controller ON

For switching off the balancing controller you set

- bit 4 to 0 Balancing controller OFF

© Siemens AG 1991 All Rights Reserved

3.2.7 Synchro-spindle

The SYNCHRO-SPINDLE function is an Option that can only be used on 32 bit servo CPUs.

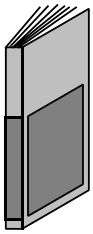
The SYNCHRO-SPINDLE function enables one leading and one following spindle (LF, FS) to rotate perfectly synchronously at an adjustable transmission ratio. This function also enables an angular offset to be programmed between the leading and following spindle.

The linear relationship between a leading spindle and the relevant following spindle is determined by the transmission ratio $K_{ü}$.

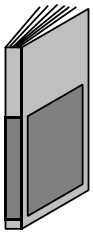
The function is selected and deselected via the G functions G201 and G202.

Up to 3 following spindles can be defined for the SINUMERIK 880.

If the SYNCHRO-SPINDLE function is selected, commands to the following spindle are ignored. If this function is not selected, the following spindle can be activated as a normal spindle.



Please refer to the ”Extended spindle functions” Function Manual for more detailed information on the functions described in this Operator's Guide.



Note:

- When the SYNCHRO-SPINDLE function is selected for a following spindle this is indicated by a setting data bit:

  SD 5004.0 for FS1
  SD 5006.0 for FS2
  SD 5008.0 for FS3

  These bits serve for indication only. They must not be changed manually.

- The functions ”Calculate $K_{ü}$ ” and ”Activate $K_{ü}$” described in this Operator's Guide can also be activated and deactivated by the PLC user program via the interface.

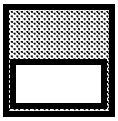
© Siemens AG 1991 All Rights Reserved

Input of transmission parameters

The transmission parameters are used to determine the transmission ratio $K_{ü}$. The rotary movement a following spindle performs with reference to the leading spindel is determined on the basis of the transmission ratio.

Sequence of operation

SETTING DATA



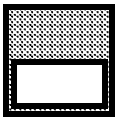
You operate the SETTING DATA softkey in any operating mode.





You extend the displayed softkey menu twice using this key.

ELG DATA



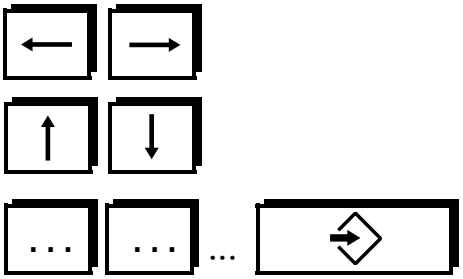
You then operate the ELG DATA softkey.

The input screen form showing the transmission parameters is displayed.

© Siemens AG 1991 All Rights Reserved

You enter the desired values for the numerator and denominator of the transmission ratio in this input screen form (see Table ”Transmission parameters”).

Proceed as follows:



You shift the input field to the desired position using the cursor keys.

You enter the relevant values using the numeric keys.

You transfer each value with the input key.

Note:

- Maximum input value 99 999 999 (8 digits)
  Max. 7 places behind the decimal point
  (e. g. 0.1234567)
- Maximum $K_{ü}$ 1000
- For the input transmission parameters to be taken into account for machining, they must first be calculated and then activated.

Setting data no.			Transmission parameters
FS1	FS2	FS3	
SD14	SD24	SD34	Numerators
SD15	SD25	SD35	Denominators

Table Transmission parameters

© Siemens AG 1991 All Rights Reserved

Calculate transmission ratios

Sequence of operation

You call the input screen form ”Common bits setting data” using the COMMON BITS softkey.

(The sequence of operation is described in Section 3.2.14.1 ”Setting data bits”.)

In this input screen form you select the SD number for the relevant following axis

- 5004 FS1
- 5006 FS2
- 5008 FS3

and set

- bit 7 to 1 Calculate new $K_{ü}$

Activate transmission ratios

Sequence of operation

You call the input screen form ”Common bits setting data” using the COMMON BITS softkey.

(The sequence of operation is described in Section 3.2.14.1 ”Setting data bits”.)

In this input screen form you select the SD number for the relevant following axis

- 5005 FS1
- 5007 FS2
- 5009 FS3

and set

- bit 7 to 1 Activate new $K_{ü}$

© Siemens AG 1991 All Rights Reserved

Switch FS overlay on/off

The setpoint suppression for the following spindle is activated when the ”FS overlay” function is switched on. This enables the following spindle to perform additional traversing motions apart from those predetermined by the leading spindle in order to establish a defined angular offset between leading and following spindle.

Sequence of operation

You call the input screen form ”Common bits setting data” using the COMMON BITS softkey.

(The sequence of operation is described in Section 3.2.14.1 ”Setting data bits”.)

In this input screen form you select the SD number for the relevant following axis

- 5004 FS1
- 5006 FS2
- 5008 FS3

and set

- bit 2 to 1 FS overlay ON

For switching off FS overlay you set

- bit 2 to 0 FS overlay OFF

© Siemens AG 1991 All Rights Reserved

Enable/disable emergency retraction

With coupled spindles any faults in synchronism may result in damage to the workpiece, the tool and the machine itself. Such damage can be largely avoided or minimized if the coupled spindles can be very quickly separated in the event of failure using suitable precautions (Emergency retraction).

Sequence of operation

You call the input screen form ”Common bits setting data” using the COMMON BITS softkey.

(The sequence of operation is described in Section 3.2.14.1 ”Setting data bits”.)

In this input screen form you select the SD number for the relevant following axis

- 5004 FS1
- 5006 FS2
- 5008 FS3

and set

- bit 5 to 1 Enable emergency retraction

If emergency retraction is not to be enabled you set

- bit 5 to 0 Emergency retraction not enabled

Note:

The ”Enable emergency retraction” function should be switched off while changeover is effected to a new transmission ratio (Set bit 5 to 0).

© Siemens AG 1991 All Rights Reserved

Acceleration limitation synchronous/not synchronous

The position synchronism of the leading and following spindles may be lost when they are accelerated. When the "Acceleration limitation synchronous" function is active, synchronism is reestablished at the end of the acceleration process.

Sequence of operation

You call the input screen form ”Common bits setting data” using the COMMON BITS softkey.

(The sequence of operation is described in Section 3.2.14.1 ”Setting data bits”.)

In this input screen form you select the SD number for the relevant following spindle

- 5004 FS1
- 5006 FS2
- 5008 FS3

and set

- bit 6 to 1 Acceleration limitation synchronous

If the ”Acceleration limitation synchronous” function is not to be activated, you set

- bit 6 to 0 Acceleration limitation not synchronous

© Siemens AG 1991 All Rights Reserved

Switch balancing controller on/off

When coupling is activated, the balancing controller should be switched on to ensure perfect synchronism in the event of malfunction.

Sequence of operation

You call the input screen form ”Common bits setting data” using the COMMON BITS softkey.

(The sequence of operation is described in Section 3.2.14.1 ”Setting data bits”.)

In this input screen form you select the SD number for the relevant following spindle

- 5004 FS1
- 5006 FS2
- 5008 FS3

and set

- bit 4 to 1 Balancing controller ON
- bit 4 to 0 Do not enable emergency retraction

© Siemens AG 1991 All Rights Reserved

3.2.8 Setting data

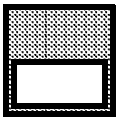
Using the **setting data** you determine certain operating states.

You can **input**, **modify** or **delete** setting data using screen forms in the control.

ANGLE OF ROTATION setting data

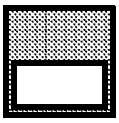
Sequence of operation for ANGLE OF ROTATION setting data

SETTING DATA



In any operating mode, activate the SETTING DATA softkey.

ZERO OFFSET

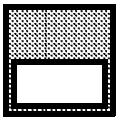


Then operate the ZERO OFFSET softkey.



Expand the softkey menu displayed by using this key, which is on the right below the CRT.

ANGLE OF ROTATION



Operate the ANGLE OF ROTATION softkey.

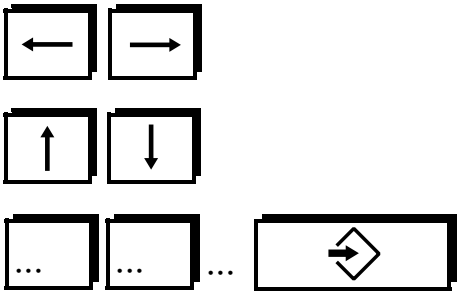
((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-HC01-0BA0
Seite 3–45 einmontieren))

In **AUTOMATIC** mode, the input screen form COORDINATE ROTATION SETTING DATA appears.

Enter the desired angles of rotation A...
for G54 to G57 in this input screen form (the angles of rotation A...
for G58 and G59 are pre-programmed).

© Siemens AG 1991 All Rights Reserved

Proceed as follows:



Using the cursor keys, shift the input field to the desired position

(G54...G59)

Using the numerical keyboard, enter a value for the angle of rotation "A" selected.

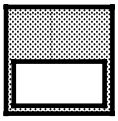
The value is then transferred to the input field by operating the input key.

© Siemens AG 1991 All Rights Reserved

SCALE CHANGE setting data

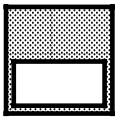
Sequence of operations for SCALE CHANGE setting data

SETTING DATA



In any operating mode, operate the SETTING DATA softkey.

SCALE



Operate the SCALE CHANGE softkey.

((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-HC01-0BA0
Seite 3–47 einmontieren))

In the **AUTOMATIC** mode, the CRT display

SCALE CHANGE SETTING DATA appears.

The following pre-programmed values are displayed:

- The "scale factor" for one channel
- The "scale centre" (reference point) for one axis

© Siemens AG 1991 All Rights Reserved

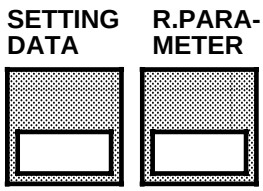
R PARAMETER setting data

The 880 control recognises parameters R0 to R599 and R900 to R999:

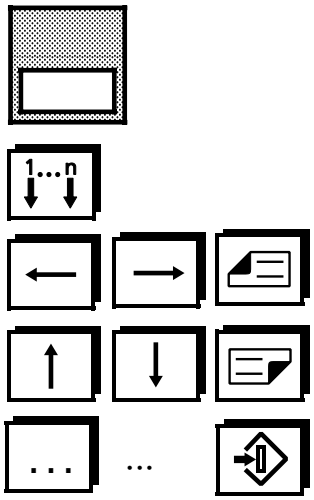
- Parameters R0...R599 are channel-specific, i.e. they exist for each channel separately.
- Parameters R900...R999 are jointly valid for all channels (central parameters).

Only certain R parameters are available for the user. The following overview shows the parameter assignments.

Sequence of operation for R PARAMETER setting data



((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-0HC01-0BA0
Seite 3–48 einmontieren))



Select either channel-specific or central R parameters with the respective softkey.

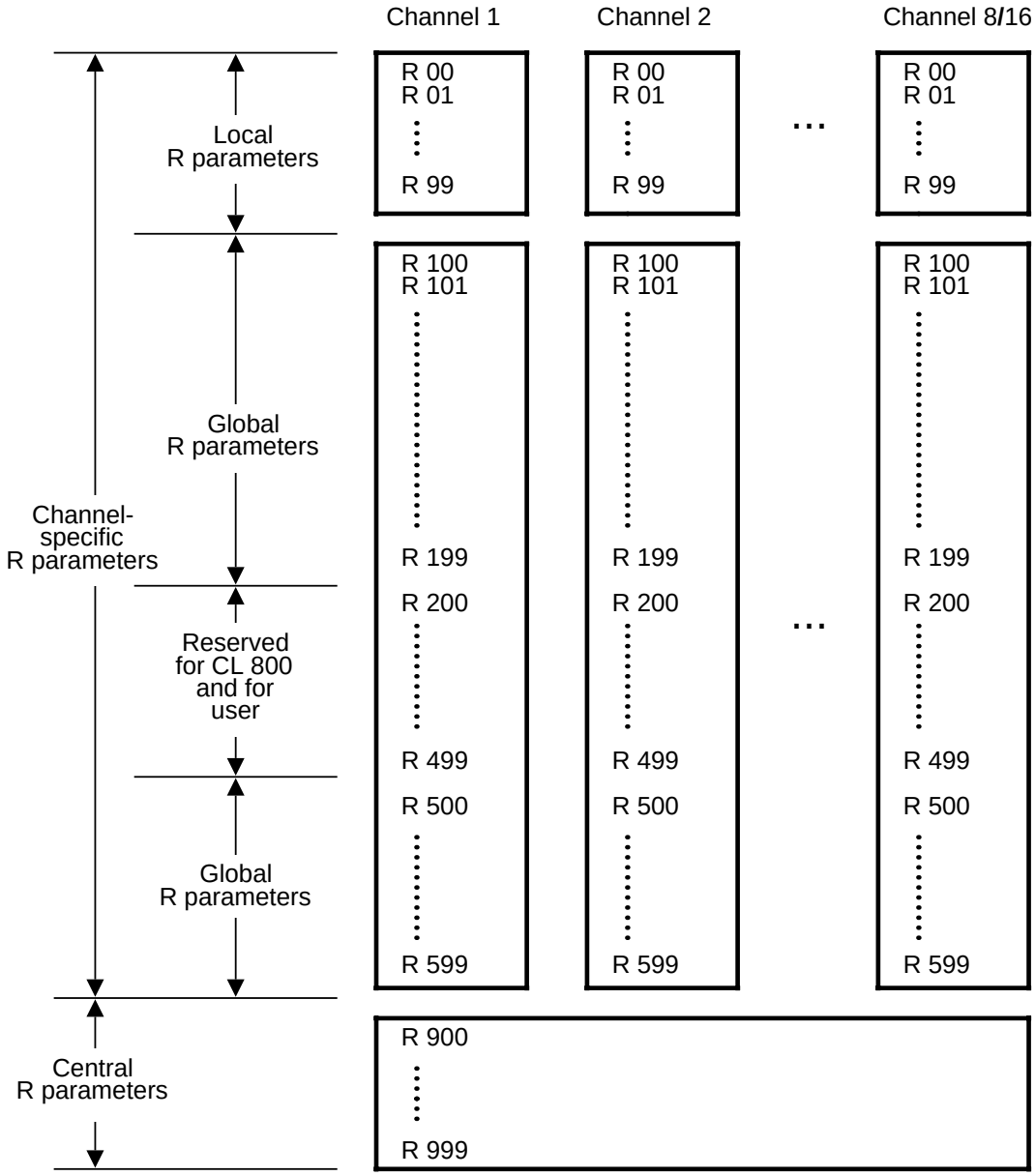
After selecting an R parameter, enter the desired value by operating the numeric keys. It is then possible to transfer the value into the input field using the yellow input key.

© Siemens AG 1991 All Rights Reserved

Parameter structure

A total of 600 parameters per channel and 100 central parameters are available:

- Parameters R 0...R 599 are channel-specific.
- Parameters R 900...R 999 are valid for all channels.



Channel-specific R parameters			
No.	Used in Cycle	Permanently assigned?	Function
R 0 : R 49	In all Siemens cycles	As long as cycles are being processed	Transfer parameters. See Descriptions: Standard cycles, Measuring cycles, Mould making cycles
R 50 : R 99	In all Siemens cycles	As long as cycles are being processed	Local R parameters. Cycle calculations are carried out with these R parameters
R 100 : R 109	L 940 (L799) : L 944	Yes	Global R parameters. Tool management
R 110 : R 199	L 898 : L 979	Yes	Global R parameters. Measuring cycle machine data
R 200 : R 219	In all Siemens cycles	As long as cycles are being processed	Converter (PG 675 or WS 800) SINUMERIK 3/8 800 converted cycles
R 220 : R 239	All cycles generated with WS800	As long as cycles are being processed	WS 800 compiler
R 240 : R 299	L960	Yes	Transfer of 3 ZO groups and 5 axes (L960 with R parameters) (Option)
R 300 : R 499	All cycles generated with WS800 and with @040 to @043	As long as cycles are being processed	WS 800 compiler (STACK for R parameters)
R 500 : R 599			Global R parameters Free for user

Central R parameters			
No.	Used in Cycle	Permanently assigned?	Function
R 900 : R 999			Free for user
R 9n : R9n+9	L 968 L 969	As long as cycles are being processed	Transfer program n = 00...90 (between working channel R110-R199 and store channel, 10 R parameters required. The number block is freely selectable.

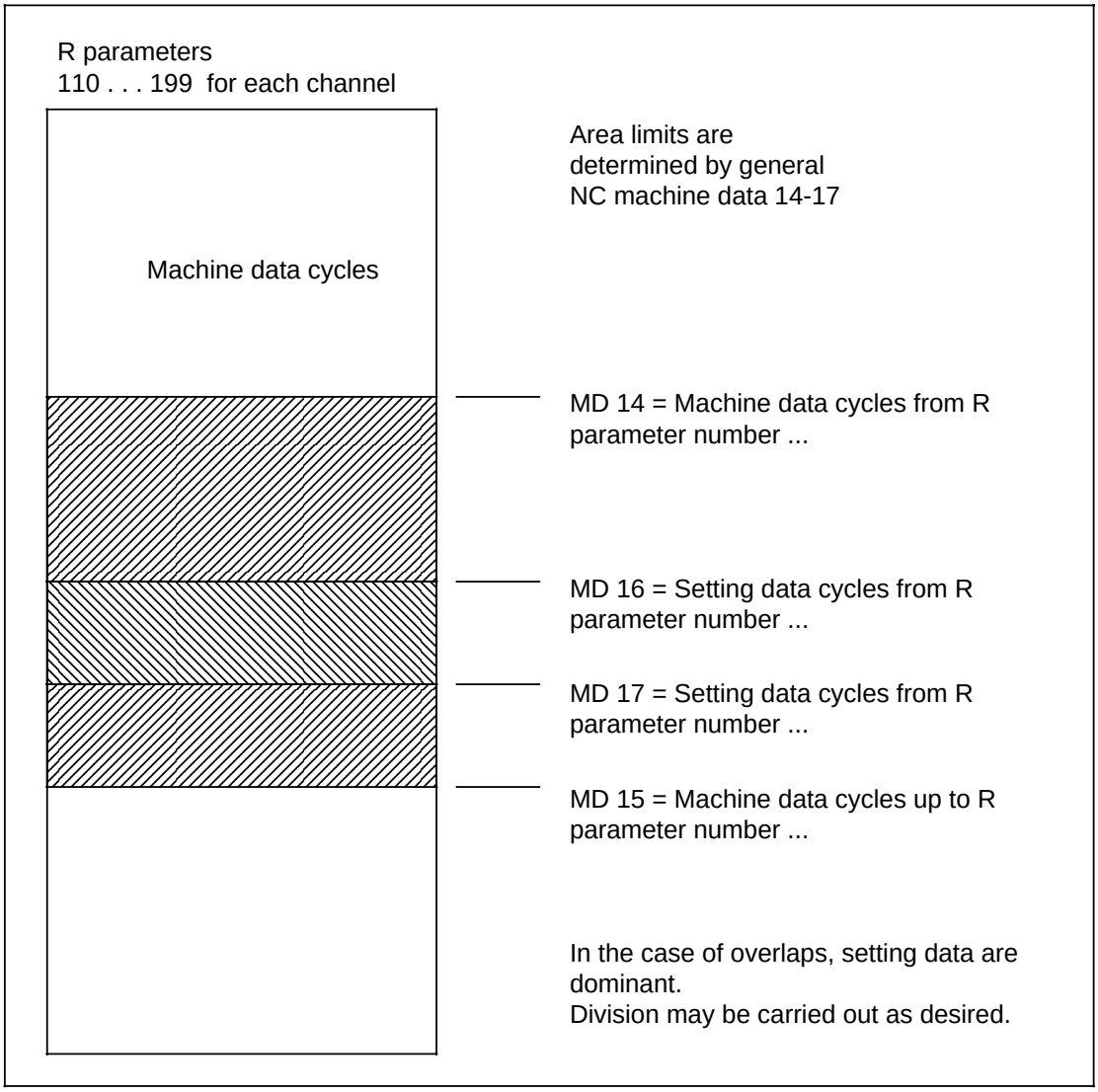
© Siemens AG 1991 All Rights Reserved

Definition of R parameters as machine data cycles and setting data cycles

So that parameters used for measuring cycles (R 110 to R 199) are protected from being modified inadvertently, parameters may be defined as

- machine data cycles or
- setting data cycles.

General NC machine data 14 to 17 are used for the definition.



The display is selected using the MACHINE DATA CYCLES or SETTING DATA CYCLES softkeys.

In simulation, there are no channel-specific machine data cycles or setting data cycles for the simulation channels.

An NC channel is allocated to each simulation channel. Cycles processed in simulation access the data of the NC channel allocated to the simulation channel.

@ acts as a general lock on writing machine data and setting data cycles in simulation.

© Siemens AG 1991 All Rights Reserved

SPINDLE setting data

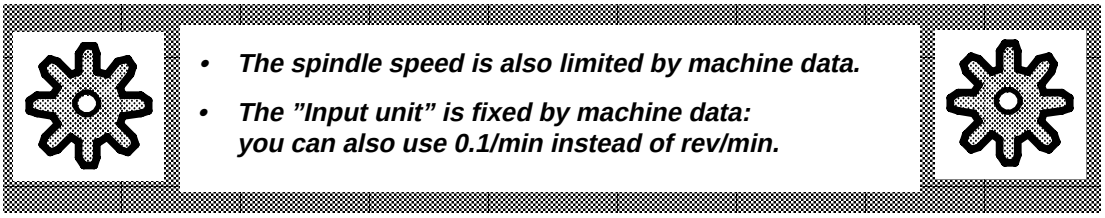
Using an input screen form, you can enter/modify the following spindle data via SETTING DATA:

a) For spindles on a 16 bit servo CPU[c)]

Designation	Standard setting data (delivery state)	Maximum input value	Input unit
Programmed spindle speed limitation	0	12 000	rev/min
Oriented spindle stop	0	359.9	1/10 degr.
Spindle speed limitation (absolute)	100	12 000	rev/min

b) For spindles on a 32 bit servo CPU

Designation	Standard setting data (delivery state)	Maximum input value 1)	Maximum input value 2)	Input unit
Programmed spindle speed limitation	0	30 000	99 999	rev/min
Oriented spindle stop	0	359.99	–	1/100 deg.
Spindle speed limitation (absolute)	100	30 000	99 999	rev/min

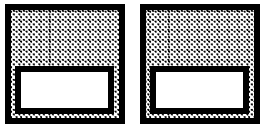


1) with actual value acquisition
2) without actual value acquisition
c) up to and including Software Version 6

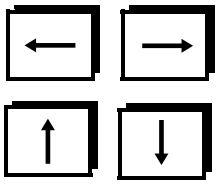
© Siemens AG 1991 All Rights Reserved

Sequence of operation for SPINDLE setting data

SETTING DATA **SPINDLE**



((Mitgeliefertes Foto
Nr. 3–49 einmontieren))



With the cursor keys, position the cursor at the desired input field.

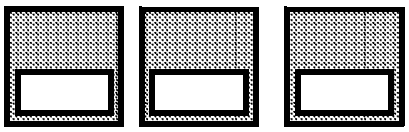
For each input field selected you can enter one value. This is then transferred to the input field when you press the input key.

You can alter an angle which has already been programmed to cut a multiple thread via the softkey START ANGLE.

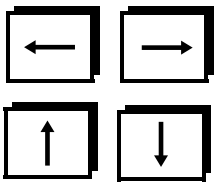
© Siemens AG 1991 All Rights Reserved

The function EXTENDED THREAD PACKAGE
(multiple thread cutting)
is an option.

SETTING DATA **SPINDLE** **START ANGLE**



((Mitgeliefertes Foto
Nr. 3–50 einmontieren))



You move the input field to the desired position using the cursor keys.

You can enter one value for every selected input field. This value is then accepted by the input field by pressing the input key.

Values < 0 and 360 degrees are calculated modulo 360 degrees. The input resolution is 0.001 degrees.

Example:

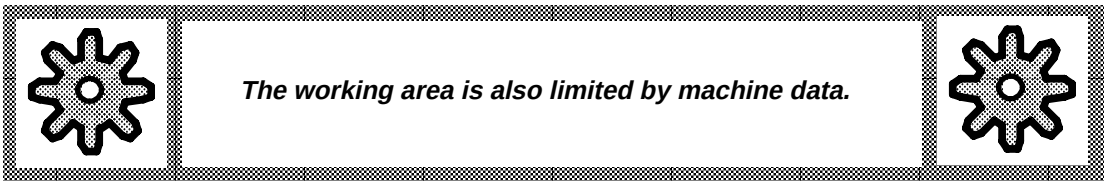
In the case of a triple thread, the part program is first executed with a 0 degree angle, then with a 120 degree angle and finally with a 240 degree angle.

© Siemens AG 1991 All Rights Reserved

AXIAL setting data

Using an input screen form, you can enter/modify the following data via “SETTING DATA“:

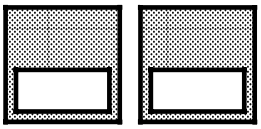
Designation	Standard setting data (delivery state)	Maximum input value	Input unit
Min. working area limit	0	+/-99999999	units (Ref. system: IS)
Max. working area limit	0	+/-99999999	units (Ref. system: IS)



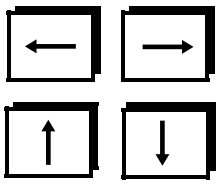
The working area is also limited by machine data.

Sequence of operation for AXIAL setting data

SETTING DATA **AXIAL**



((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-0HC01-0BA0
Seite 3–54 einmontieren))



With the cursor keys, position the cursor at the desired input field.

© Siemens AG 1991 All Rights Reserved

COMMON setting data

You can enter/modify the following data using an input screen form:

- Smoothing constant with threading
- Dry run feedrate

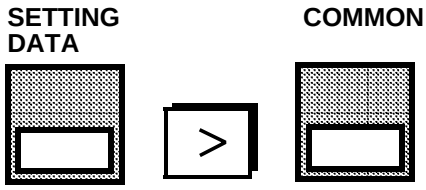
Designation	Standard setting data (delivery state)	Maximum input value	Input unit
Smoothing constant with threading	0	5	1)
Dry run feedrate	0	4 900 000	1000 units (Ref. system: IS)

1) The speed ramp and smoothing time is obtained from the input value according to the following formula:

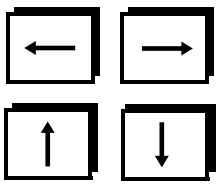
$$2^{\text{Input value}} - 1 \cdot \text{IPO clock}$$

The IPO clock is determined via a machine data and ranges between 10 and 30 ms.

Sequence of operation for COMMON setting data



((Mitgeliefertes Foto Nr. 3–52 einmontieren))

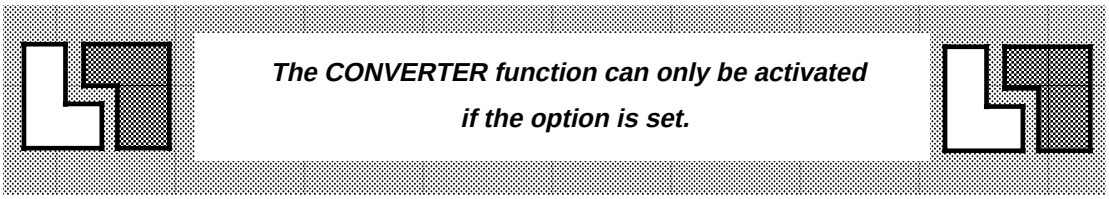


You shift the input field to the desired position using the cursor keys.

You can enter one value for each input field selected. This value is then transferred to the input field using the input key.

© Siemens AG 1991 All Rights Reserved

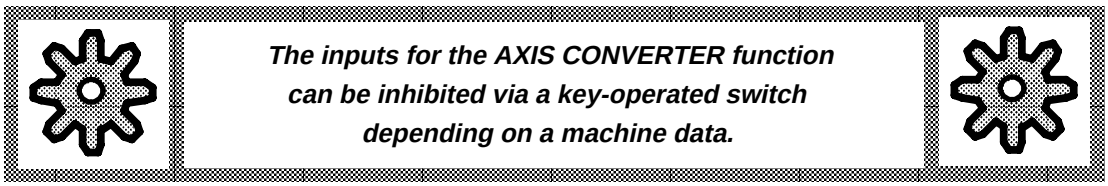
CONVERTER setting data



The axis/spindle addresses specifed in a part program are converted to other axis/spindle addresses when a converter is active. This enables part programs to be executed on a machine on which the specified addresses are differently defined or not defined at all.

The SPINDLE CONVERTER function can only be activated via the PLC. In other words, there is no way for the user to activate this function via the operator panel.

The AXIS CONVERTER function is activated by entering it in the channel-specific setting data display CONVERTER. The assignment of the programmed axis to the converted axis is likewise entered in this display.



The two functions AXIS CONVERTER and SPINDLE CONVERTER are only active in the AUTOMATIC or MDI AUTOMATIC mode. Conversion is not active during overstore.

With current block display the programmed axes and spindles are shown.

A more detailed description of the converter functions is given below.

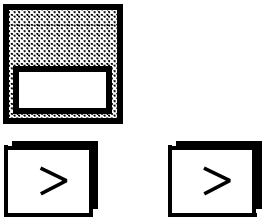
AXIS CONVERTER

Sequence of operation

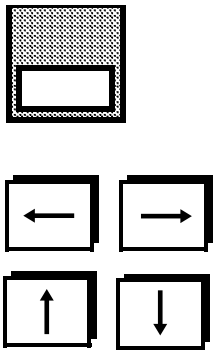
The control is in one of the operating modes **AUTOMATIC** or **MDI AUTOMATIC**.

© Siemens AG 1991 All Rights Reserved

SETTING DATA



CONVERTER



((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-0HC01-0BA0
Seite 3–57 einmontieren))

With the cursor keys you can move the input field to the desired position.

You can specify a value for every input field selected. The value is then taken over into the input field when you press the enter key.

You can input up to eight conversion pairs per channel.

A format check is performed on the input. You can input:

- a letter or
- a letter and one or two numbers (address extension)

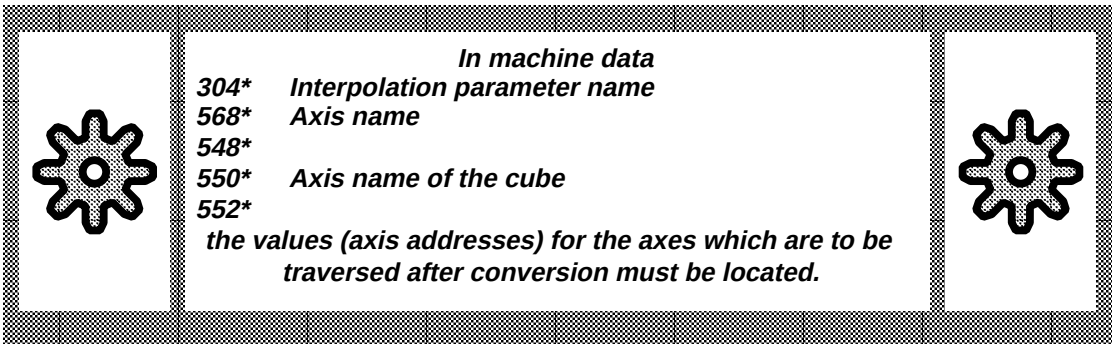
You can also make the following conversion

- X=Z1
- Y=Z1

If in the part program the axis addresses X and Y are in different blocks. If X and Y are programmed in one block, alarm No. 3011 "Axis programmed twice" is displayed.

You can activate the conversion function in **AUTOMATIC** and **MDI AUTOMATIC** modes.

© Siemens AG 1991 All Rights Reserved



The conversion setting data (e. g. X13 = X14) can neither be read nor written via the @ 310, 311, 410, 411.

The conversion status active yes/no can be read or written via @ 311, 312, 411, 412.

The conversion setting data with the exception of the conversion status cannot be read in or out via the V.24 interface. Transformation axes cannot be converted.

SPINDLE CONVERTER

The values specified by the PLC are displayed in a channel-specific setting data display (see AXIS CONVERTER). The assignment between the programmed spindle and the machine spindle can be modified by the PLC while the part program is being executed. This requires the running part program to be synchronised with the PLC user program via a M function.

The user cannot modify the values specifed by the PLC via the operator panel.

Note:

- A maximum of 3 spindles can be converted simultaneously for each NC channel.
- Conversion is not active during simulation.
- The address extensions of the spindle-specific M function (M03, M04, M05 and M19) are likewise converted accordingly. It should be noted that only the address extension is converted. If a spindle is a leading spindle in one of the channels, this leading spindle as well must be converted.

© Siemens AG 1991 All Rights Reserved

3.2.9 Program input

You can enter a program

- using the keyboard
- using operator prompting

You can switch from one mode to the other while entering programs, i. e. you can **alternate** between operator prompting and the keyboard.

3.2.9.1 Program input with the keyboard

Only the keyboard is used to enter character by character an opened program %... with blocks N... L_F.

Sequence of operation

Select one of the following operating modes with the mode selector switch:



PRESET (Set actual value)



JOG (Feed/Jog)



AUTOMATIC (Automatic mode)

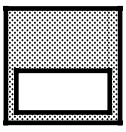
1, 10, 100,
1 000, 10 000

INC FEED 1...10 000 (Incremental feed)



REFPOINT (Approach reference point)

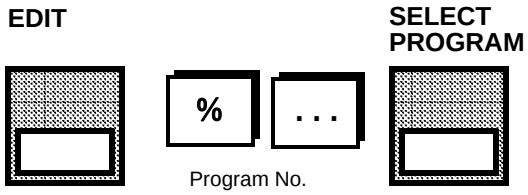
PART
PROGR.



((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-HC01-0BA0
Seite 3–59 einmontieren))

© Siemens AG 1991 All Rights Reserved

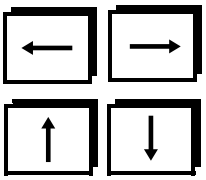
The control displays the **last** part program you called up some time previously, MPF 77 in this example.



You can now enter a new program in the screen form displayed or modify an existing program.

Each word of a newly entered block (including the block number and the "L_F" character for the end of the block) must be concluded with the yellow input key.

The following keys are available in addition to the address and numerical keys:



These keys enable you to **position** the cursor at any point: in front of a block or in front of any word within a block.



This key is used to **delete** a whole block or a word behind the cursor.



This key is used to **modify** a whole block or a word behind the cursor.

Apart from program input, the PART PROGRAM

function can also be used for

- ***program editing (see Section 3.2.11)***
- ***program management (handling, see Section 3.2.12 for overview)***
- ***program simulation (see Section 3.2.13)***

© Siemens AG 1991 All Rights Reserved

3.2.9.2 Program input with operator prompting

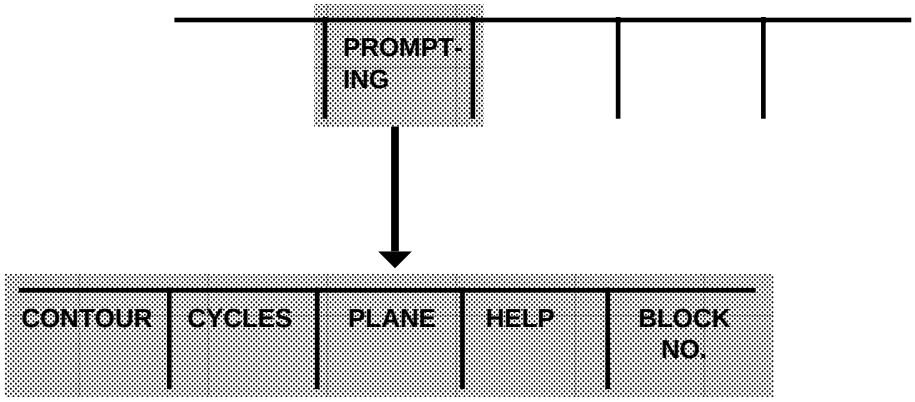
Program input **with** operator prompting means:

You enter an opened program with the **aid of a menu**:

Input functions which often recur are offered as a menu.

Using the menu saves you time and effort and considerably reduces the possibility of errors.

You call up the operator prompting function by calling the PROMPTING function. When you press the respective softkey, the following functions (displayed in the screen and softkey menu; see below) are available:



The menu tree for operator prompting is shown in detail in Section 2.4.9.

© Siemens AG 1991 All Rights Reserved

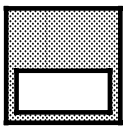
3.2.9.3 Plane selection

Sequence of operation

Operate the PLANE softkey.

The following display appears on the CRT:

PLANE



((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-HC01-0BA0
Seite 3–62 einmontieren))

After entering the channel number by way of the keyboard and selecting the plane G17, G18 or G19 using a softkey, the control automatically fills the abscissa, ordinate and applicate fields and the two interpolation parameters.

The STORE PLANE softkey is used to transfer the selected plane into the part program. In addition, the axes are correctly specified in the input displays of the CONTOUR and CYCLES functions.

When plane G16 is selected, the values marked in the input display are entered using the keyboard. Where axes have an extended address, "=" is automatically added behind the address extension, so that it is recognized as such by the decoding.

The STORE PLANE softkey is used to transfer the plane G16 into the part program. The axes are then additionally entered in the part program.

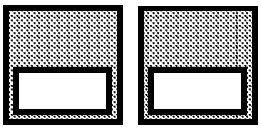
© Siemens AG 1991 All Rights Reserved

3.2.10 Contour

You can select contour elements with the CONTOUR function. Graphics displays on the CRT provide support during programming.

Sequence of operation

PROMP-TING **CONTOUR**



((Foto aus der Druckschrift SINUMERIK 880, Bedienungsanleitung, Bestellnr. 6ZB5 410-HC01-0BA0 Seite 3–63 oben einmontieren))



((Foto aus der Druckschrift SINUMERIK 880, Bedienungsanleitung, Bestellnr. 6ZB5 410-HC01-0BA0 Seite 3–63 unten einmontieren))

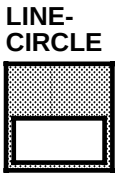


The previous display reappears.

You can now select a softkey function from the menu displayed at the bottom of the screen.

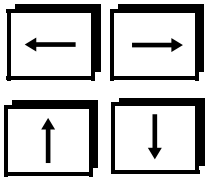
© Siemens AG 1991 All Rights Reserved

Now select the contour element you require by pressing the respective softkey, e.g. the LINE-CIRCLE function:

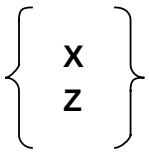


((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-HC01-0BA0
Seite 3–64 einmontieren))

Now enter the geometric values for the contour in all the input fields one after the other and conclude each entry with the input key.



You can use the cursor keys to jump to any input field, for example if you wish to change a geometric value you have already entered for the selected contour.



Note:

In input fields where several values in inverse video are enclosed in curly brackets ("X" and "Z" in this example).

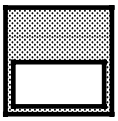
You may only enter one value (i.e. in this example, either the value for "X" or for "Z").

If you enter more than one value and press the "STORE" softkey, the ”BFM MORE THAN ONE VALUE” error message appears in the interactive dialog line on the display.

BFM=Operator prompting screen form.

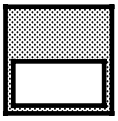
© Siemens AG 1991 All Rights Reserved

DELETE PARAM.



Press the DELETE PARAMETER softkey if you have entered more than one value by mistake.

STORE



Press the STORE softkey when you have entered all the values required for the contour.

The contour definition is now stored in the part program with all geometric values entered. The control automatically generates the L_F (end of block) character and displays the complete block entered.

© Siemens AG 1991 All Rights Reserved

3.2.11 Program correction (EDIT): Insert/modify/delete word, insert/delete block

You can edit any part program stored in the program memory.

Program correction can also be used while another part program is being processed.

The following corrections are possible:

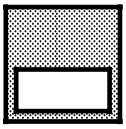
- Input word
- Modify word
- Delete word
- Input block
- Delete block

Sequence of operation

The control is in one of the following operating modes:

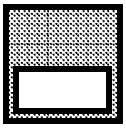
PRESET/JOG/AUTOMATIC/INC FEED 1...10 000/REFPOINT

PART PROGR.



((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-HC01-0BA0
Seite 3–66 einmontieren))

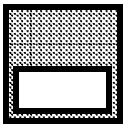
EDIT



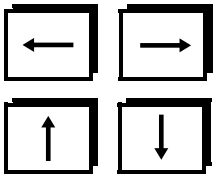


Programm No. [1])

SELECT PROGRAM



You can now either modify an existing part program or enter a new program in the displayed screen form.



Using the cursor keys, set the cursor, marked in inverse video, in front of the point to be edited.

[1])"%" for main program or "L" for subroutine

© Siemens AG 1991 All Rights Reserved

Insert word



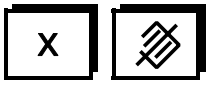
Enter the address **and** the value ("X 15" in example) and store it using the input key.

Modify word



Enter the address **and** the **new** value ("X 25" in example) and press the "Modify word" key.

Insert block



Enter the address of word to be deleted ("X" in example) and press the "Delete word/block" key.

Delete word

N5 .....L_F

Position the cursor **in front of** the block which will **follow** the inserted block (in the example a new block will be inserted in front of block "N 5...").

N4 .....L_F

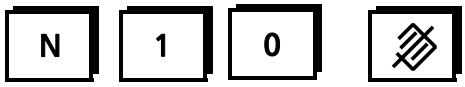
Now enter the block to be inserted ("N 4" in the example).

Each word of the newly entered block is concluded by pressing the input key. The "L_F" key marks the "end of the block".

© Siemens AG 1991 All Rights Reserved

Delete block

N10 .....L_F

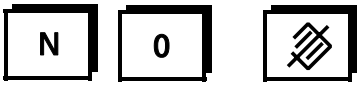


Position the cursor **in front of** the block to be deleted ("N 10" in the example).

Key in the block number and operate the "Delete word/block" key.

For blocks without a block number, proceed as follows:

G .....F.....X.....L_F



Position the cursor **in front of** the first word in the block to be deleted (in the example, preparatory function "G...").

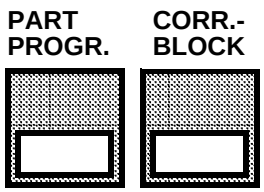
3.2.11.1 Correction display (CORRECTION BLOCK)

During processing or simulating (see Section 3.2.13) a program, program processing is stopped and the relevant alarm displayed if the control recognises a programming error. Using the CORRECTION BLOCK function the block or the word to be corrected is marked.

Sequence of operation

The control is in one of the following operating modes:

PRESET/AUTOMATIC/INC FEED 1...10 000/JOG/REFPOINT. The program has been interrupted and an alarm displayed.



The cursor is now positioned in front of the block/word recognised as containing an error. You can now make the correction without changing the setting.



When you have corrected the relevant address and entered the appropriate value, operate the "Modify word" key.



After the correction operate this key to continue program processing (not possible with alarms which must be acknowledged by pressing the RESET key).

© Siemens AG 1991 All Rights Reserved

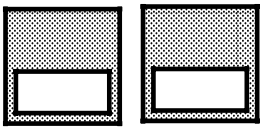
3.2.12 Program management

3.2.12.1 Displaying the stored programs (DIRECTORY)

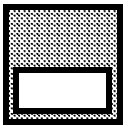
Sequence of operation

The control is in one of the following operating modes:

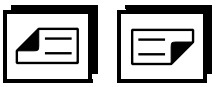
PRESET/JOG/AUTOMATIC/INC FEED 1...10 000/REFPOINT



((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-HC01-0BA0
Seite 3–69 einmontieren))



The CRT display is switched over to the directory. The main programs, subroutines or cycles are shown according to the softkey used.



Pressing the "Page up**/**down" keys causes further programs or cycles to be displayed.

Since data other than main programs and subroutines can also be stored in the part program memory, all the identifiers consist of at most three letters and one following number. For main programs this is "MPF" and for subroutines or cycles it is "SPF". The identifiers "%" for main programs and "L" for subroutines and cycles can also be used.

The total memory available for part programs is divided into sectors with a fixed size of 520 bytes.

The directory displays both the actual length of a part program and the memory space used up by it. The free memory space is indicated both in characters and in sectors.

Memory space which becomes free because of deletions is immediately made available again as free memory.

© Siemens AG 1991 All Rights Reserved

3.2.12.2 Protection of subroutines (Cycle lock)

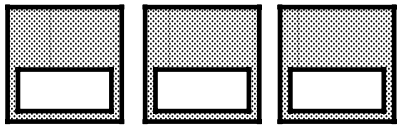
You can protect subroutines from unauthorised output or inadvertent deletion by locking them:

Sequence of operation

The control is in one of the following operating modes:

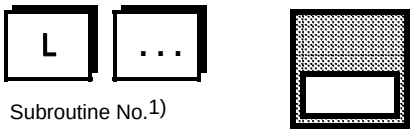
PRESET/JOG/AUTOMATIC/INC FEED 1...10 000/REFPOINT

PART PROGR. **DIREC-TORY** **SUB-ROUT.**



The subroutine directory appears on the screen.

LOCK



Subroutine No.[1)]

The control enters the protected subroutine in the cycle list. Cycles are protected subroutines.

1) *L or SPF*

© Siemens AG 1991 All Rights Reserved

3.2.12.3 Cycle lock release (ENABLE)

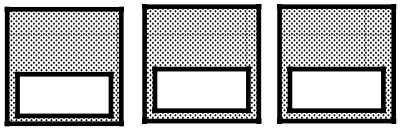
It is not possible to release the lock on fixed cycles which are stored on the EPROM.

Sequence of operation

The control is in one of the following operating modes:

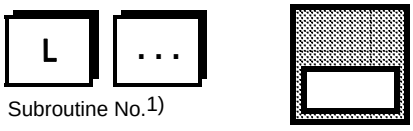
PRESET/JOG/AUTOMATIC/INC FEED 1...10 000/REFPOINT

PART PROGR. **DIREC-TORY** **CYCLES**



The directory of cycles (protected subroutines) appears on the screen.

ENABLE



Subroutine No.[1)]

The control enters the cycle in the list of (unprotected) subroutines.

The cycles can be protected against unauthorised output by programming the PLC accordingly.

This effectively protects the machine tool manufacturer's or user's knowhow.

It is then no longer possible for the operator to enable and disable programs.

1) *L or SPF*

© Siemens AG 1991 All Rights Reserved

3.2.12.4 Copying a program (COPY)

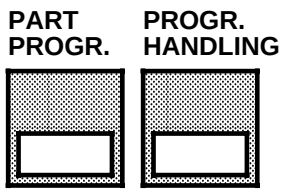
Using the COPY function, you can store the same program under different program numbers in the program memory. This enables you to keep one program with the contents unchanged and make changes to the copy.

Also, by copying a program with COPY, you can compress programs which have been greatly fragmented by changes and require too much storage space. To allow this function to be used sensibly, the program overview display shows not only the actual size of a program but also the amount of space it occupies in memory.

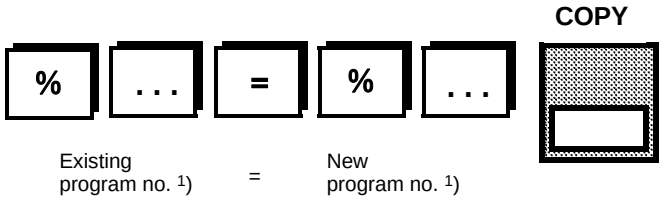
Sequence of operation

The control is in one of the following operating modes:

PRESET/JOG/AUTOMATIC/INC FEED 1...10 000/REFPOINT



((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-HC01-0BA0
Seite 3–72 einmontieren))



Existing program no. [1)] = New program no. [1)]

The control now copies the program with the existing program no. and also stores it with the new.

---

1) *or "L . . ." for subroutines; for "%" and "L" also MPF or SPF can be used.*

© Siemens AG 1991 All Rights Reserved

3.2.12.5 Moving cycles (MOVE)

If the cycle lock is released by the PLC, it is possible to copy a cycle from the EPROM to the program memory using the MOVE function. This cycle can then be edited like a part program.

Since there are now two cycles with the same number "SPF" ... in different locations (in the EPROM and in the program memory), the cycle is identified by an asterisk (*) in the EPROM to distinguish it in the directory. On selection, the cycle in the program memory has priority.

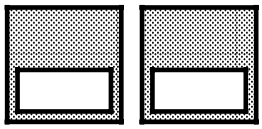
The cycle in the EPROM can only be called up again using the SELECT PROGRAM softkey if the corresponding cycle is deleted from the program memory.

Sequence of operation

The control is in one of the following operating modes:

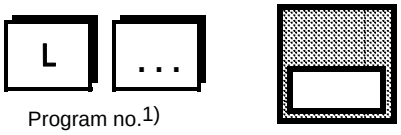
PRESET/JOG/AUTOMATIC/INC FEED 1...10 000/REFPOINT

PART PROGR. **PROGR. HANDLING**



((Foto aus der Druckschrift SINUMERIK 880, Bedienungsanleitung, Bestellnr. 6ZB5 410-HC01-0BA0 Seite 3–72 einmontieren))

MOVE



Program no.[1)]

The program selected is copied from the EPROM to the part program memory.

[1)] *L or SPF*

© Siemens AG 1991 All Rights Reserved

3.2.12.6 Simultaneous editing of programs

In order that programs in the part program memory can be edited simultaneously they must be protected against being executed.

Several programs may be protected against being executed. The identifier "#" in the part program directory indicates which program is to be edited simultaneously with program execution.

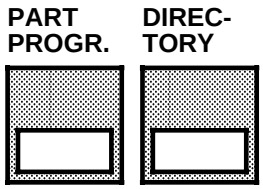
If an attempt is made to inhibit a part program just being processed, the message ”Only possible in Reset” is output.

Programs which are newly created are always locked and must be released for execution using the appropriate softkey function.

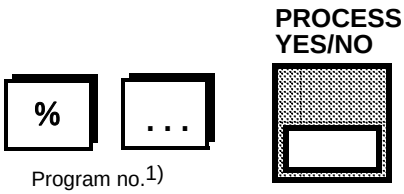
Sequence of operation

The control is in one of the following operating modes:

PRESET/JOG/AUTOMATIC/INC FEED 1...10 000/REFPOINT



The program directory appears on the CRT.



The selected program is either locked or released for execution.

1) % or MPF

© Siemens AG 1991 All Rights Reserved

3.2.12.7 Renaming a program (RENAME)

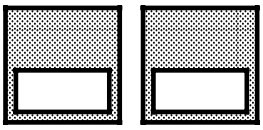
Using the RENAME function, you can change a program number.

Sequence of operation

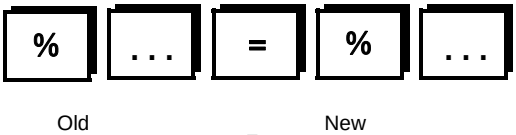
The control is in one of the following operating modes:

PRESET/JOG/AUTOMATIC/INC FEED 1...10 000/REFPOINT

PART PROGR. **PROGR. HANDLING**



((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-HC01-0BA0
Seite 3–72 einmontieren))

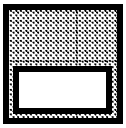


Old program no. [1])

=

New program no. [1])

RENAME



The control changes the part program number. The program itself remains unchanged.

1) % or MPF, or L or SPF

© Siemens AG 1991 All Rights Reserved

3.2.12.8 Deleting a program (DELETE)

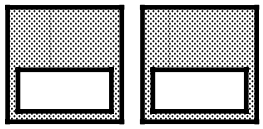
Using the DELETE function you can either delete each program singly or delete a sequence of programs.

Sequence of operation

The control is in one of the following operating modes:

PRESET/JOG/AUTOMATIC/INC FEED 1...10 000/REFPOINT

PART PROGR. **PROGR. HANDLING**

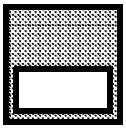


((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-HC01-0BA0
Seite 3–72 einmontieren))



Program no.[1)]

DELETE



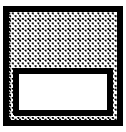
The program with the number entered is deleted by the control.

or



From program no. [1)] = To program no. [1)]

DELETE



All programs from "%..." to "%..." inclusive are deleted by the control.

Cycles must be enabled prior to being deleted.

[1)] *% or MPF, or L or SPF*

© Siemens AG 1991 All Rights Reserved

3.2.12.9 Reorganizing the program memory

The part program memory is automatically reorganised after a part program has been deleted.

The table below illustrates the relationship between the size of the part program memory and the maximum number of programs which can be managed.

Memory size	Maximum no. of programs	Assigned sectors			Sectors
		Programs	Directory	MDI	
32 K	55	55	4	2	61
64 K	115	115	7	2	124
96 K	174	174	10	2	186
128 K	234	234	13	2	249
256 K	470	470	27	2	499

Here, a maximum of 560 characters is assumed for each part program.

© Siemens AG 1991 All Rights Reserved

3.2.12.10 Selecting a program

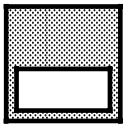
Using the SELECT PROGRAM function, you can either create a program or call a program already stored in the program memory, e.g. for editing.

Sequence of operation

The control is in one of the following modes of operation:

PRESET/JOG/AUTOMATIC/INC FEED 1...10 000/REFPOINT

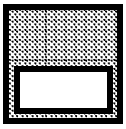
PART PROGR.



((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-HC01-0BA0
Seite 3–78 einmontieren))

The control displays the part program which you called the **last** time you used the control, in this example program "MPF 77".

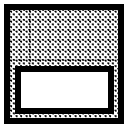
EDIT



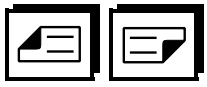
% **. . .**

Program no. [1])

SELECT PROGRAM



The program number called is displayed on the CRT together with the start of the program and the first blocks if there is a program with this number already in the memory.



With the "Page up/down" keys, you can call up the parts of the program not yet displayed (if already in memory).

[1]) *% or MPF, or L or SPF*

© Siemens AG 1991 All Rights Reserved

3.2.13 Simulation

The SIMULATION function is an option.

With the SIMULATION function, it is possible to test a program. The traversing movements of the tool are simulated graphically on the CRT.

3.2.13.1 Simulation data

- A graphics display with input fields provides a support for the input of the simulation data.

The simulation window determines which part of processing is to be simulated. It thus sets the size of representation.

Simultaneous simulation (simulation in parallel with processing) is possible by means of machine data.

Please note:
It is possible that the NC channels and the simulation channel can be mutually influenced by common data.

Common data are, for example:
- ***Settable zero offsets***
- ***Programmable zero offsets***
- ***Tool compensations***
- ***Global R parameters***
- ***Machine data***
- ***Setting data***

Any influencing of the data in the NC channels by the simulation channel can be prevented using an inhibit. Simulation simultaneously with processing is not then possible.

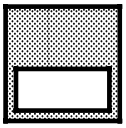
© Siemens AG 1991 All Rights Reserved

Sequence of operation

The control is in one of the following modes of operation:

PRESET/JOG/AUTOMATIC/INC FEED 1...10 000/REFPOINT

**SIMULA-
TION**

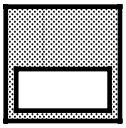


Press the SIMULATION softkey.
The basic display for simulation appears on the CRT.

((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-HC01-0BA0
Seite 3–80 oben einmontieren))

Input of simulation data:

**SIMUL.
DATA**



Press SIMULATION DATA softkey.
Depending on the type of control, one of the following displays will appear on the CRT:

SINUMERIK 880 M

((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-HC01-0BA0
Seite 3–80 unten einmontieren))

© Siemens AG 1991 All Rights Reserved

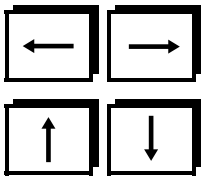
SINUMERIK 880 T

((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-HC01-0BA0
Seite 3–81oben einmontieren))

SINUMERIK 880 T with twin-saddle graphics

((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-HC01-0BA0
Seite 3–81unten einmontieren))

The current input field is shown in inverse video.
By inputting the program number the program to be simulated is selected. Simulation data is determined by inputting the appropriate values or symbols in the tables on the CRT.



These keys can be used to position the cursor to any input field.

Each value input is stored on operation of the yellow input key and is then displayed on the CRT in the input area selected.

© Siemens AG 1991 All Rights Reserved

The simulation data is used to determine the following functions, depending on the type of control:

- **Type of representation**
  Any modification only becomes active when simulation is restarted.
- **Plane**
  Selection of the plane in which simulation is to take place.

  With the T/TT version, this input is omitted since only one plane is permissible.

  The plane is initialized on input of the NC channel number.
- **Settable zero offset**
  Selection of whether the settable zero offset is to be taken into account with simulation.

  Modifications become effective on restarting simulation.
- **Programmable zero offset**
  Selection of whether the programmable zero offset is to be taken into account with simulation.

  Modifications become effective on restarting simulation.
- **Skippable block**
  Selection of whether skippable blocks are to be taken into account.
  Modifications are taken into account in the current simulation after a delay.
  The selection and deselection in double-saddle simulation applies to both simulation channels. The skip levels must not be controlled via the PLC in simulation.
- **Programmed stop**
  Selection of whether the programmed stop (M01) is to be taken into account.
  Modifications are taken into account in the current simulation after a delay.
- **Single block**
  Selection of whether simulation is to operate in single-block mode.

  Modifications are taken into account in the current simulation after a delay.
- **Draw in rapid traverse**
  Selection of whether the tool paths are to be drawn in rapid traverse blocks too.
- **Delete paths on tool change**
  Selection of whether the tool paths previously drawn are to be deleted on tool change. This means that representation of the tool paths of the active tool is not affected by previous tracks.
- **NC channel number**
  Specification of which NC channel the program to be simulated will later run in. This means that the data of the NC channel selected (plane reset position, tool data, working area, etc.) is used for simulation.
- **Simulation area (normal display)**
  Input of simulation area.

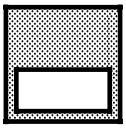
  Make sure that the values are within the working area limits.

  The values become active for simulation when the NORMAL DISPLAY softkey is operated.
- **Block search**
  Input of block search indicator, similar to NC block search.

© Siemens AG 1991 All Rights Reserved

3.2.13.2 Input of simulation tool data

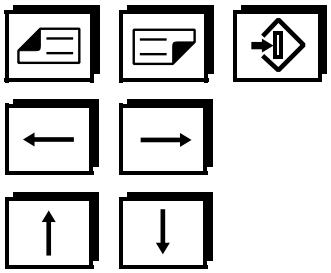
TOOL DATA



Press the TOOL DATA softkey.
The following display appears on the CRT.

((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-HC01-0BA0
Seite 3–83 einmontieren))

The input screen form displayed is used to aid input of the "Simulation tool data". The following keys are available in addition to the numeric ones:



These keys are used to select the desired position number.

These keys can be used to position the cursor to any input field.

Each value input is stored on operation of the yellow input key and is then displayed on the CRT in the input field selected.

The position number is in front of the tool number. It indicates the memory position of a simulation tool data block.

The tool number "T" determines which data block is used for tool representation.

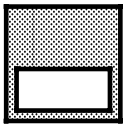
The tool compensation number "D" is the cross reference between the simulation tool data and the NC tool data (input by means of the TOOL OFFSET softkey). This enables you to check that simulation tool data and NC tool data agree.

© Siemens AG 1991 All Rights Reserved

Input of the tool holder values does not depend on the tool.

The values of tool data can also be input with the aid of graphic operator support.

INPUT GRAPHICS



Press the INPUT GRAPHICS softkey.
Depending on the type of control, one of the following displays appears on the CRT:

SINUMERIK 880 M

((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-HC01-0BA0
Seite 3–84 oben einmontieren))

SINUMERIK 880 T

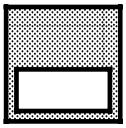
((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-HC01-0BA0
Seite 3–84 unten einmontieren))

The values for tool data are entered by means of the keyboards and the input key.

The NC tool data (TO data) determined by the tool number "D" are displayed.

© Siemens AG 1991 All Rights Reserved

UPDATE DISPLAY



Press the UPDATE DISPLAY softkey.

The tool is illustrated with the values input on the right-hand side of the CRT.

The tool edge from the NC tool data is inserted on the right-hand display.

(◆).

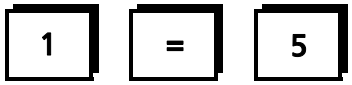
Simulation tool data and NC tool data are thus compared.

The tool edge inserted must agree with the tool edge in the graphic display.

In case of disagreement, please check the simulation tool data. If the simulation tool data is not feasible a question mark is displayed.

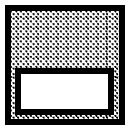
It is possible to copy the simulation tool data. This allows you to determine similar tools of different dimensions very quickly, since only the differing dimensions have to be input on the copy.

Sequence of operation



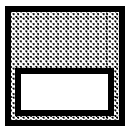
Input of position numbers using numeric and alpha keyboards.

COPY



After pressing the COPY softkey, the tool data block in memory position 1 is copied to memory position 5.

SADDLE

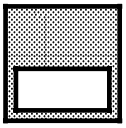


The SADDLE softkey only has significance for switching over tool data blocks for twin-saddle lathe (control type TT).

© Siemens AG 1991 All Rights Reserved

Operator prompting for inputting the tool shape:

TOOL SHAPE



Press the TOOL SHAPE softkey.
Depending on the type of control, one of the following displays appears on the CRT:

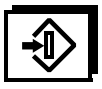
SINUMERIK 880 M

((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-HC01-0BA0
Seite 3–86 oben einmontieren))

SINUMERIK 880 T

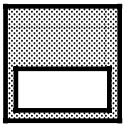
((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-HC01-0BA0
Seite 3–86 unten einmontieren))

The display gives you an overview of the definition of tool shapes. The appropriate characters can be input in this display as four digits.

Example: DLLA  .

© Siemens AG 1991 All Rights Reserved

TOOL SELECT



The TOOL SELECTION softkey is used to select operator prompting for input of the tool shape.

1st example: Lathe tool (SINUMERIK 880 T)

((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-HC01-0BA0
Seite 3–87 oben einmontieren))

2nd example: Milling cutter (SINUMERIK 880 M)

((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-HC01-0BA0
Seite 3–87 unten einmontieren))

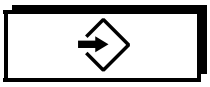
© Siemens AG 1991 All Rights Reserved

The tool shapes are illustrated in detail with operator prompting.

Please input the appropriate character for the position indicated top right in the display in inverse video in the input line.

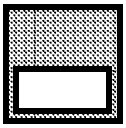


Press the RECALL key after completing input (4 characters).



The values are stored and displayed in the "Tool shape" display once the yellow input key is operated.

INPUT LIST



The INPUT LIST softkey gives you the possibility of jumping to the "Tool data list" display.

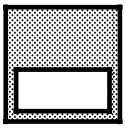
© Siemens AG 1991 All Rights Reserved

3.2.13.3 Simulation data (BLANK)



After the jump back to the ”Simulation data” display, press the BLANK softkey.

BLANK



Depending on the type of control, one of the following displays appears on the CRT:

SINUMERIK 880M

((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-HC01-0BA0
Seite 3–89 oben einmontieren))

SINUMERIK 880T

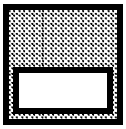
((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-HC01-0BA0
Seite 3–89 unten einmontieren))

Now input the appropriate dimensions for the BLANK into the input screen form, one input field at a time (marked in inverse video) through the keyboard. Conclude each input by pressing the input key. The inverse video then jumps to the next field.

© Siemens AG 1991 All Rights Reserved

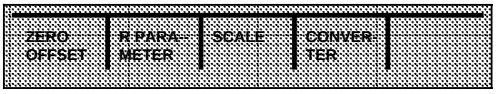
3.2.13.4 Simulation data (SETTING DATA)

SETTING DATA

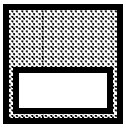


Press the SETTING DATA softkey.

You now have the choice of the following softkey functions:

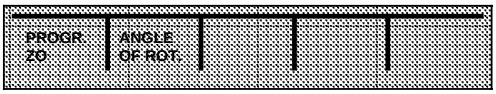


ZERO OFFSET

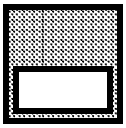


Press the ZERO OFFSET softkey.

You now have the choice of the following softkey functions:



PROGR. ZO



Press the PROGRAMMABLE ZERO OFFSET softkey.

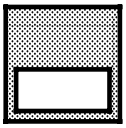
The following display appears on the CRT:

((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-HC01-0BA0
Seite 3–90 einmontieren))

The values of programmable zero offsets are displayed.

© Siemens AG 1991 All Rights Reserved

ANGLE OF ROT.



Press the ANGLE OF ROTATION softkey.

The following display appears on the CRT:

((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-HC01-0BA0
Seite 3–91 oben einmontieren))

Only with two-saddle graphics

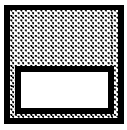
The required values G54...G57 are input by way of the numeric keyboard and the input key.



The RECALL key allows you to jump back into the next menu up.

Press the R PARAMETER softkey.

R PARA-METER



The following display appears on the CRT:

((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-HC01-0BA0
Seite 3–91 unten einmontieren))

Only with two-saddle graphics

The required values are input by way of the numeric keyboard and the input key.

© Siemens AG 1991 All Rights Reserved

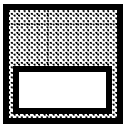


The RECALL key allows you to jump back into the next menu up.

Press the SCALE softkey.

The following display appears on the CRT:

SCALE



((Foto aus der Druckschrift SINUMERIK 880, Bedienungsanleitung, Bestellnr. 6ZB5 410-HC01-0BA0 Seite 3–92 einmontieren))

Only with two-saddle graphics

The values for scale change are displayed.

© Siemens AG 1991 All Rights Reserved

You can execute the conversion function in simulation as well.

The conversion data can only be activated if the option is set.

You input the conversion data, as described, in a special channel-specific setting data display.

SIMULA-TION

SIMUL. DATA

SETTING DATA

CONVERTER

SINUMERIK 880T/880TT

((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-HC01-0BA0
Seite 3–93 einmontieren))

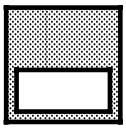
© Siemens AG 1991 All Rights Reserved

3.2.13.5 Sequence of simulation



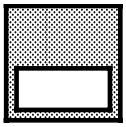
The RECALL key allows you to jump back into the simulation basic display.

START



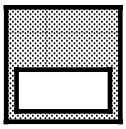
Press the START softkey.
Simulation axis traversing begins.

STOP



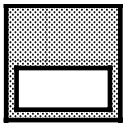
The STOP softkey allows you to stop the simulation sequence.

RESET



By pressing the RESET softkey, simulation is returned to the starting position.

START SEARCH



The block search defined in the simulation data display is activated by pressing the START SEARCH softkey.

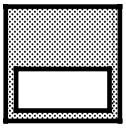
The control begins simulation at the block selected. All the blocks in the program which are in front of the selected block number are ignored.

A simulation RESET is triggered when you leave the simulation display (during a simulation).

© Siemens AG 1991 All Rights Reserved

3.2.13.6 Selection of simulation window

DISPLAY



Press the DISPLAY softkey.

Depending on the type of control, one of the following displays appears on the CRT:

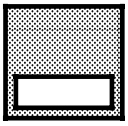
SINUMERIK 880 M

((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-HC01-0BA0
Seite 3–95 oben einmontieren))

SINUMERIK 880 T

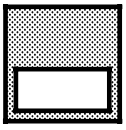
((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-HC01-0BA0
Seite 3–95 unten einmontieren))

NORMAL DISPLAY



By pressing the NORMAL DISPLAY softkey, you accept the values input into the "Simulation data" display as the simulation window.

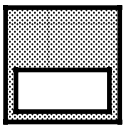
FULL DISPLAY



The FULL DISPLAY softkey allows you to choose the working area limitation values as the simulation window.

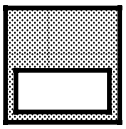
© Siemens AG 1991 All Rights Reserved

ZOOM +

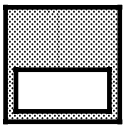


The ”ZOOM +” and ”ZOOM – ” softkeys allow you to decrease or increase the values of the current simulation window.

ZOOM

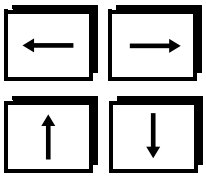


SET CURSOR



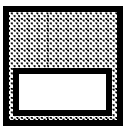
A variable simulation window can be set by positioning the cursor (X) to opposite corner points of a window.

Sequence of operation

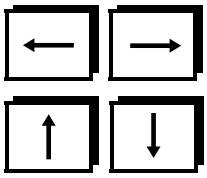


The first corner point is determined using the cursor keys.

SET CURSOR

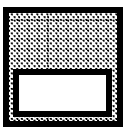


The previously defined corner point is set using the SET CURSOR softkey.



The second corner point is positioned using the cursor keys.

SET CURSOR



The window specified by the two corner points is set using the SET CURSOR softkey.

© Siemens AG 1991 All Rights Reserved



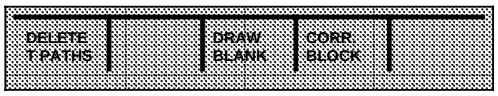


Press the RECALL key

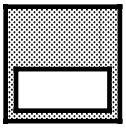
and

then the "ETC" key.

The following functions are now available.

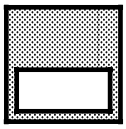


DELETE T PATHS



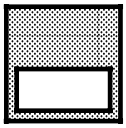
This key is used to clear the tool paths at any point in time.

DRAW BLANK



By operating the DRAW BLANK softkey the blank is drawn again during simulation.
This may be required in several planes in one machining operation.

CORR. BLOCK



The CORRECTION BLOCK softkey is used to locate a program error.
After it is operated, the cursor goes to the position in front of the wrongly programmed word.

If errors occur in several channels at the same time, then the one in the channel with the lowest program number is displayed.

© Siemens AG 1991 All Rights Reserved

3.2.13.7 Example of simulating a part program for SINUMERIK 880M

```
%16
N1 G1 Z50 F100 LF
N2 X20 Y10 D3 LF
N3 Z10 LF
N4 L1 P5 LF
N5 M30 LF
```

Subroutine L1

```
N1 G1 G91 Y50 F500 LF
N2 X10 LF
N3 Y-50 LF
N4 X10 LF
N5 M17 LF
```

((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-HC01-0BA0
Seite 3–98 einmontieren))

The following are displayed on the screen:

%16	Part program number	**N . .**	Subroutine block number
N . .	Block number	**Y . ., X . .**	Simulation plane with area specified
L . .	Subroutine number	**X . ., Y . ., Z . .**	Current axis values
P . .	Number of remaining passes	**F . .**	Current feedrate

Note:

- Mirror-image machining is not possible.
- If the feedrate (F) is not specified in the program, e. g. during simulation of subroutines, the dry run feedrate is taken as the feedrate value for simulation with linear interpolation with G01 or with circular interpolation with G02**/**G03.

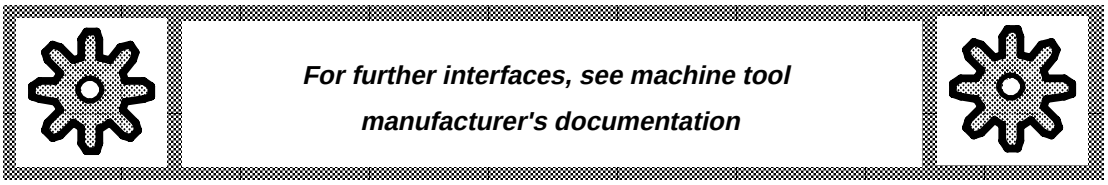
  The alarm 3051 "Prog. feed missing**/**wrong" is given to indicate that the feed value is missing. The alarm has no effect on simulation and can be acknowledged with the key "Clear alarm".

© Siemens AG 1991 All Rights Reserved

3.2.14 Data input/data output

For the input and output of data you will find an RS232C (V.24)**/**20 mA interface (= **Interface 1** with variable device connections possible) on the front of the machine operating panel.

A further RS232C (V.24) interface (=**Interface 3**, with fixed assigment for the connection of a particular device) is located at the back of the control.



Fixed interface data is defined for the interfaces. This data can be modified by calling the SETTING DATA function.

Modification of the interface data – especially for interface 1 – is necessary for example when a different device is interfaced to the SINUMERIK 880 (printer, tape reader, programmer units, etc.).

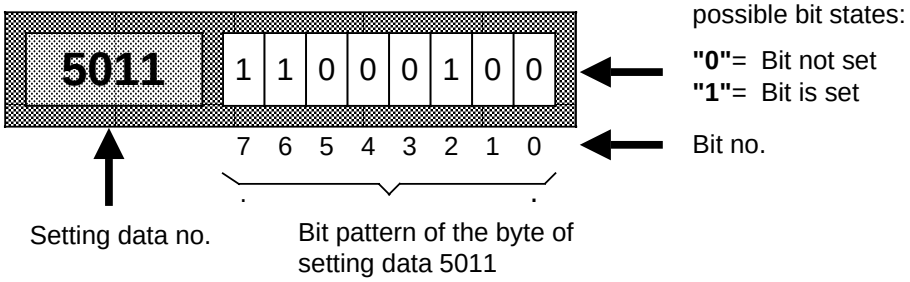
You will find the necessary information for setting/modifying data bits in Section 3.2.14.1 (Sequence of operation) and Section 6.3 (Assignment of the setting data for connection of peripheral devices).

3.2.14.1 Setting data bits

The setting data - from byte no. 5000 - each consists of 8 bits: no. 0 to no. 7. This setting data serves for the matching of interfaces for the connection of different devices.

Explanation of setting data structure:

(e.g. for setting data (SD 5011)



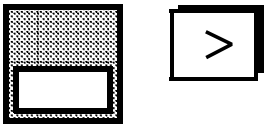
Using an input screen form on the CRT you can set the desired bit pattern to any byte in the range from 5000 on!

© Siemens AG 1991 All Rights Reserved

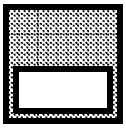
Sequence of operation

The control is in any operating mode:

SETTING DATA



((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-HC01-0BA0
Seite 3–100 einmontieren))

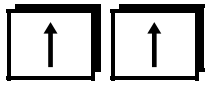


Select the setting data range you require using a softkey.



Using the cursor keys, you can position the cursor to any other setting data

or



using the keyboard you enter the number of the required setting data and operate the search key.



The input field marked in inverse video is now positioned on the 8-bit pattern of the setting data called. Using the "0" or "1" numerical keys, enter the desired 8-bit pattern, and store the entire sequence of characters using the input key. Leading zeros (”0”) need not be entered in the bit pattern.

Continue in this manner until all the required items of setting data have been set.

© Siemens AG 1991 All Rights Reserved

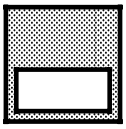
3.2.14.2 Data input

Sequence of operation

The control is in one of the following operating modes:

PRESET/JOG/AUTOMATIC/INC FEED 1....10 000/REFPOINT

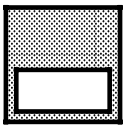
DATA IN/OUT



((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-HC01-0BA0
Seite 3–101 einmontieren))

The control gives you the identifier of the device type to be connected for each interface (for identifiers, see Section 6.3).

INTERFACE 1+2 . . .

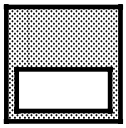


((Mitgeliefertes Foto
Nr. 3–98 einmontieren))

© Siemens AG 1991 All Rights Reserved

Select one of the interfaces displayed with the numeric keys and the input key.

START INPUT



Selecting this you enable the control's receive function. On the upper right-hand side of the CRT the message **DIO** (Data input/output) is displayed.

Under **Data type**: the data coming from the transmission device (tape reader, programmer unit...) is identified.

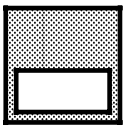
You enter the program type (main program=1, subroutine=2) and the program numbers of the program sequence to be transferred with the numeric keypad and the enter key.

Note:

If a connection already exists to a remote PG (via a SINEC H1 LAN), the dialog text, "Remote PG running/stop ?" appears when the softkey INPUT START has been pressed. You must carry out the following steps to terminate the connection:

- Press REMOTE PG STOP softkey
  The connection to the remote PG is terminated and the dialog text message "Remote PG is inactive" appears.
- Press INPUT START softkey
  The local PG is selected.

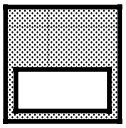
STOP



With this softkey you can interrupt data input for the selected interface. The **DIO** display is extinguished.

Data input is resumed when you press the START INPUT softkey again.

STOP ALL



This softkey interrupts data input for all interfaces. Data input can only be resumed under certain conditions.

© Siemens AG 1991 All Rights Reserved

The table below explains the type of data stored in the SINUMERIK 880 (see also 880 Programming Guide):

Data Type	Meaning
MPF	Part program (**M**ain **P**rogram **F**ile)
SPF	Subroutine (**S**ub **P**rogram **F**ile)
PCF	PLC error message texts (**PL**C **F**iles)
TOA	Tool compensation (**T**ool **O**ffset **A**ctive)
ZOA	Zero offset (**Z**ero **O**ffset **A**ctive)
TEA1	NC machine data (**Te**sting Data **A**ctive 1)
TEA2	PLC machine data (**Te**sting Data **A**ctive 2)
TEA3	IAR machine data (**Te**sting Data **A**ctive 3)
TEA4	Machine data cycles
RPA	R parameters with assigned values (**R P**arameter **A**ctive)
SEA	Addresses with assigned values (**Se**tting Data **A**ctive)
SEA4	Setting data cycles
CLF	Clear instruction (**Cl**ear **F**ile)
UMS	**U**ser **M**emory **S**ubmodule
TOG	Simulation tool data (**T**ool **O**ffset **G**raphics)
IKA1	IKA relations (interpolatory compensation with absolute values)
IKA2	Error curve pointers (interpolatory compensation with absolute values)
IKA3	Compensation points (interpolatory compensation with absolute values)

Please note:
During data input it is not possible for the control to call up specific data.

© Siemens AG 1991 All Rights Reserved

Note:

When inputting from punched tape, the input data is checked for simple errors.
All characters in punched tape code have a common identification character:

- in ISO code there is always an even hole count
- in EIA code there is always an odd hole count

The control automatically identifies the correct code when reading the first "%" (ISO) or "EOR" (EIA)!

The "Odd hole count" or ”Even hole count" criterion is used for character parity checking from the second character in the program onwards.

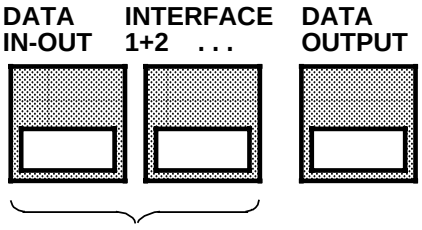
As a further check, when a program already in the program memory is read in again, a complete program comparison is carried out. When an error is recognized, reading in is stopped and the error displayed on the CRT.

3.2.14.3 Data output

Sequence of operation

The control is in one of the following operating modes:

PRESET/JOG/AUTOMATIC/INC FEED 1....10 000/REFPOINT



see also Data input

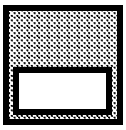
((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-HC01-0BA0
Seite 3–103 einmontieren))

© Siemens AG 1991 All Rights Reserved

Select one of the interfaces displayed with the numeric keys and the input key.



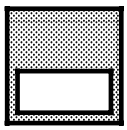
Expand the softkey menu with this softkey.



Select the data type to be output with this softkey.

Select the softkey function, for example, PART PROGRAM:

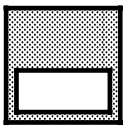
PART PROGR.



An input screen form appears into which you enter the programs and subroutines to be output.

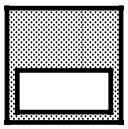
Key into the input fields marked in inverse video the numbers of the programs with which output is to begin and end. Conclude each entry with the yellow input key. The inverse marking then jumps to the next field. The cursor keys enable you to select any input field.

MAINPRG. START



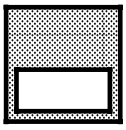
or

SUBROUT. START



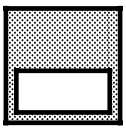
You start the control's transmit function (output of data).

STOP



Using this softkey, you can stop data output at the interface selected and continue it again from the beginning with MAINPRG. START or SUBROUT. START.

STOP ALL



This softkey interrupts data output for all interfaces.

© Siemens AG 1991 All Rights Reserved

In the same way, you can select and start the output of data for the following:

- TOOL COMPENSATION
- ZERO OFFSET
- MACHINE DATA
- PLC MACHINE DATA
- SETTING DATA
- R PARAMETER
- COMPENSATION DATA

which have not been detailed here.

Note:

- Selection of data, i.e. output via the operater panel of certain sections – as with PART PROGRAM – is not possible.
- Data output with or without leader or trailer can be set using a setting data (SD 5016 bit 1).
- The input of a channel no. is required by the control for R PARAMETERS
  – "1" to ”16” for local output, channels 1 to 16[c)]
  – ”1” bis ”8” for local output, channels 1 to 8[d)]
  – "0" for global output of R parameters
- When outputting machine data it is possible to output only machine data that are not equal to zero. This accelerates the output. When reading in data protected in this way, make sure that machine data that have not been output (MD=0) are not overwritten.

  This type of output can be selected for each interface via setting data.

c) up to and including Software Version 6

d) from GA2, Software Version 1

© Siemens AG 1991 All Rights Reserved

3.2.14.4 File transfer

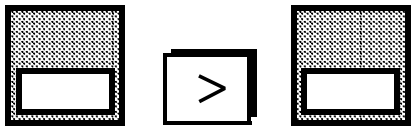
You can inititate data transfer from and to the host computer by manual operation using the FILE TRANSFER softkey.

Sequence of operation

The control is in one of the operating modes:

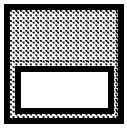
PRESET/JOG/AUTOMATIC/INC FEED 1....10 000/REFPOINT

DATA IN-OUT > **FILE TRANSFER**



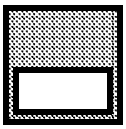
((Foto aus der Druckschrift SINUMERIK 880, Bedienungsanleitung, Bestellnr. 6ZB5 410-HC01-0BA0 Seite 3–106 einmontieren))

TRANS-FER



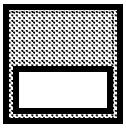
Using this softkey you transfer the values entered in the input screen form.

INPUT START



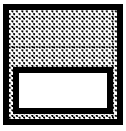
The receive function of the control is enabled. On the screen (top right) the display **DIO** (data input**/** output) appears.

OUTPUT START



The transmit function of the control is activated. On the screen (top right) the display **DIO** (data input**/**output) appears.

STOP



You stop data transfer with this softkey. The display **DIO** is cleared.

© Siemens AG 1991 All Rights Reserved

3.3 Operation

3.3.1 Starting a part program

Please note:
Before starting a part program, the control and machine must be equipped for the processing of the part program. The control must be in the Reset state.

Sequence of operation

Set the operating mode selector switch on the machine control panel to this symbol:



AUTOMATIC

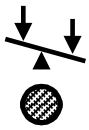
((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-HC01-0BA0
Seite 3–107 einmontieren))

Enter the desired part program number with the numeric keys and the input key.





The green "Program running" LED display lights up.

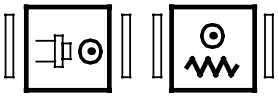


The green "Position not yet reached" LED display is lit until all axes have reached their set positions.

Note: When the red "Feed Hold" LED



display is lit, program processing is stopped.



You can enable program processing by operating the "Spindle start" key or the "Feed start" key, provided that **no program start inhibit** has been generated by the PLC (e.g. Alarm 2039).

After the program start, either ACTUAL VALUES or ACTUAL BLOCK appear on the CRT display.

© Siemens AG 1991 All Rights Reserved

3.3.2 AUTOMATIC mode
Calling the ACTUAL VALUES or ACTUAL BLOCK display

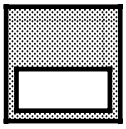
Prerequisite:

An active program is being processed in **AUTOMATIC** mode (see Section 3.3.1).

The actual values for the program running are displayed on the screen.

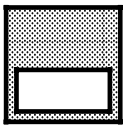
Sequence of operation

ACTUAL BLOCK



((Foto aus der Druckschrift SINUMERIK 880, Bedienungsanleitung, Bestellnr. 6ZB5 410-HC01-0BA0 Seite 3–108 oben einmontieren))

ACTUAL VALUES



((Foto aus der Druckschrift SINUMERIK 880, Bedienungsanleitung, Bestellnr. 6ZB5 410-HC01-0BA0 Seite 3–108 unten einmontieren))

© Siemens AG 1991 All Rights Reserved

Explanation of the ACTUAL VALUES display

% ...

Display of the selected program (Program pointer)

N.. L.. P.. N..

Processing pointer, indicating the block number N... and the subroutine number L..., pass count P... and the block number within the subroutine with two levels of nesting.

Prog.values Act. values
S1..... S1.....
F.....M F.....

Programmed and actual values of spindle speed and feedrate (identifier after the feedrate value:
"R"=feedrate per revolution for spindle
"M"=linear feedrate for spindle
”r”=feedrate per revolution for rotary axis;

S...%
F...%

Spindle speed override and feedrate override in %

Aux. functions
T= D H= M=

Tool number ("T"), tool offset number ("D") and further auxiliary/miscellaneous functions.

Actual pos. Dist. to go
X..... X.....
Y..... Y.....
Z..... Z.....

Actual position and distance to go value of the axes

Explanation of the ACTUAL BLOCK display

In addition to the displays already mentioned (ACTUAL VALUES), the following are displayed:

- The processing pointer with nesting of subroutines to a depth of three
- The block before the actual block
- The actual block
- The block after the actual block
- All current G functions

If the "actual block" consists of more than 39 characters, the following block is not displayed.
Likewise, a block before or after the "actual block" with more than 39 characters is not displayed.

© Siemens AG 1991 All Rights Reserved

3.3.3 Influencing AUTOMATIC operation

General:

Influencing **AUTOMATIC** operation (e.g. through programmed functions) is displayed in the first line of the CRT display.

Display: **HOLD: AUTO interrupted**

Interruption of **AUTOMATIC** operation (Program Hold), see Section 3.3.3.4 "Interrupting the program"

Display: **HOLD: Single block**

The single block has been processed (with switch position "Single block - ON" or DEC.SBL YES/NO softkey)

Display: **HOLD: Pr. stop M00, M01**

Programmed interruption of program execution.

With "Program start"  the processing of the program is continued

Display: **HOLD: Read enable**

Read enable is a PLC output signal. When the read enable signal is removed, processing of the current NC block is completed. The next program block is not transferred for processing.

Display: **HOLD Dwell time**

Execution of the program is interrupted for the duration of the programmed dwell time.

"Feed hold" is **not** displayed on the CRT, but is shown by the red LED  .

© Siemens AG 1991 All Rights Reserved

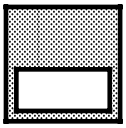
3.3.3.1 Influencing the program

With the operation of the softkeys described here active programs can be influenced.

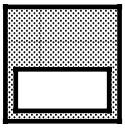
Sequence of operation

The control is in the **AUTOMATIC** operating mode

PROGRAM CONTROL



((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-HC01-0BA0
Seite 3–111 einmontieren))

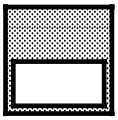


By operating any of the softkeys, you can change over the preset "NO" to "YES". A function marked with "YES" is active and is displayed on the first line of the CRT display.

© Siemens AG 1991 All Rights Reserved

Explanation of the softkeys for influencing the program

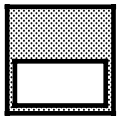
DRY RUN
YES-NO



All blocks in which a feedrate is programmed will now be traversed at the feedrate value entered into the "Dry run feedrate" setting data instead of the programmed feedrate.

The dry run feedrate then also applies instead of "Feed per revolution" G 95/G195 and the feedrate for thread cutting.

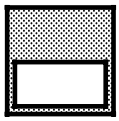
OPT.STOP
YES-NO



When an "M01" is present in the part program, the program is stopped.

When the softkey function is marked with "NO", "M01" is ignored.

CORR.
YES-NO



This signal presupposes that there is a special feedrate override switch, on the customer operating panel, for example. When the softkey function is marked with "NO", the override switch is only valid for ”Feedrate”.

When the softkey function is marked with "YES", the override set with this switch also applies to Rapid traverse.

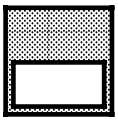
User notes on "RAPID OVERRIDE":

When installing a new NC program: the rapid override switch is released by a key-operated switch. The following rapid override values can be set with the selector switch on the customer operator panel

Setting	Override values in %
1	1
2	10
3	50
4	100

This table is applicable only when the "Rapid override switch present" machine data is set. When this is not the case, the setting for the "Feedrate override switch" applies to rapid override, with a limitation of 100%.

© Siemens AG 1991 All Rights Reserved

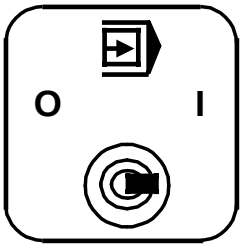


When the function is activated with "YES", the control generates the "Decoding single block active" signal.
The signal **is active at the end of the block** in the current part program which is decoding when the signal is present.

Repeatedly pressing the "Program start" key causes the decoding single blocks to be processed in sequence.

Note:

The "Single block" function works in a similar way (see Section 2.1.2).



Setting the switch to "1" activates the "Single block" function.
The generated signal is active at the end of the (current) block being processed.

The "SBL" (Single block) display appears in the first line of the CRT.

Operating the "Progr. start" key causes the single block to be processed (etc., see above). The following table indicates in which blocks the "Decoding single block active" signal or the "Single block active" signal must be present if a program is to be processed on a block-by-block basis.

Block Type	Single block	Decoding single block
Traversing blocks		
Blocks without path information		
Calculation blocks		
Switching and auxiliary function blocks		
Internally generated control blocks (CRC ...) 2)		
Threading blocks without dry run feedrate	1)	
Threading blocks with dry run feedrate		

"Single block" or "Decoding single block" signal is required

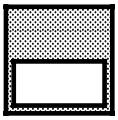
1) Single block: The stop is made at the end of the first non-threading block

2) with cutter radius/tool nose radius compensation

- Blocks which have been "pre-processed" in the buffer memory without the "Decoding single block" signal present but which have not yet been processed cannot be stopped.
- A "Decoding single block" can be influenced using OVERSTORE (see Section 3.3.3.2).

© Siemens AG 1991 All Rights Reserved

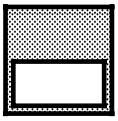
DRF
YES-NO



With activated DRF (Differential Resolver Function) it is possible to activate the handwheel for an axis to be selected (see Section 3.3.9).

The handwheel axis assignment is defined via PLC selection.

1..8
YES-NO



Blocks in the program which are marked with an **oblique** before the block number ("**/**N...") are skipped.

The skip levels are differentiated by marking the skip blocks with "**/**1" ... "**/**8".

If the character "**/**" (oblique) is placed in front of the skip block without a number, the block is skipped when "**/**" (skip block) is selected.

When one of the nine skip levels is selected the status display SKIP appears.

The individual skip levels allow you to create a complex program in a simple way. For example, you can deselect I**/**O devices not required (loaders, sensors etc.) in setup mode.

© Siemens AG 1991 All Rights Reserved



((Foto aus der Druckschrift SINUMERIK 880, Bedienungsanleitung, Bestellnr. 6ZB5 410-HC01-0BA0 Seite 3–115 oben einmontieren))



((Foto aus der Druckschrift SINUMERIK 880, Bedienungsanleitung, Bestellnr. 6ZB5 410-HC01-0BA0 Seite 3–115 unten einmontieren))

Modality in successive skip blocks.

The modality applies to blocks that are processed in decoding.

Example:

```
%1
    N5
/   N10
/   N15
/   N20
/   N25
    N30
    N35
/   N40
    N45
```

Skip "**/**" was selected when program %1 is started.

If the skip level is deselected, for example, between blocks N15 and N25, skipping blocks is kept active until the skip identifier of a block changes.

In example %1 this means that blocks N20 and N25 will still be skipped. With block N30 the deselection becomes active for the block preparation sequence because the block N30 is not marked with a skip criterion.

The block N40 is not skipped.

© Siemens AG 1991 All Rights Reserved

Example: `%2`

```
    N5
/   N10 G X Y  LF
/1  N15
/1  N20
/1  N25
/1  N30
/2  N35
/2  N40
/3  N45
/1  N50
    .
    .
    .
```

When program "%2" is started skip levels "**/**1" and "**/**2" are activated.

Between blocks N15 and N20 skip level "**/**1" is deselected.

Because of their modality blocks N20, N25, N30 are still skipped.

Because of the skip identifier "**/**2" of block N35 the modality is cancelled for "**/**1" and activated for "**/**2".

Because skip "**/**2" is also selected blocks N35 and N40 are also skipped. Blocks from N45 are traversed.

Within a group of skip blocks inactive states are also maintained.

Example: `%3`

```
/1  N5 ..
/1  N10 ..
/1  N15
/1  N20
    N25
/1  N30
/1  N35
    N40
    .
    .
    .
```

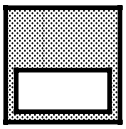
If, in the example "%3", skip level "**/**1" is selected between blocks N10 and N15, the blocks from N30 are skipped.

© Siemens AG 1991 All Rights Reserved

3.3.3.2 Execution from external

The EXECUTION FROM EXTERNAL function is an Option.

PROC. EXTERN.



Using this function you can read in and simultaneously execute part programs from an external mass storage via serial interfaces or via the computer link.

The function is selected by pressing the EXECUTION FROM EXTERNAL softkey and the function is deselected by pressing this key again.

When this function is active, the program to be executed is selected in the AUTOMATIC basic display and started with ”Start program”. Execution is started as soon as the selected program is being read into the memory. This is displayed on the 1st CRT line with the EXT message. Program parts that have been executed are deleted to clear the memory for new program parts (so-called circular buffer).

Execution can be interrupted with ”Stop program” and resumed with ”Start program”.

Execution is concluded with RESET, the associated interface is stopped and the remainder of the executed program is deleted.

If the read-in program is processed faster than new program parts are being read in, it may happen that the memory runs empty. This results in workpiece machining being stopped and the ”No block available” error message appears. The problem can be avoided by a general reduction of the feedrate.

The problem of the memory being emptied can also be avoided by filling it up completely before time-critical program sections are processed.
This can be achieved, for instance, by special M functions in conjunction with the PLC user program.

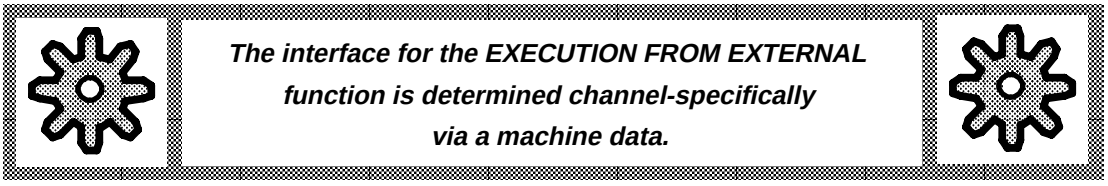
© Siemens AG 1991 All Rights Reserved

When the CURRENT BLOCK DISPLAY function is active only the current and possibly the next block are displayed on the CRT because the preceding block may already have been deleted. No display will appear when execution is extremely fast.

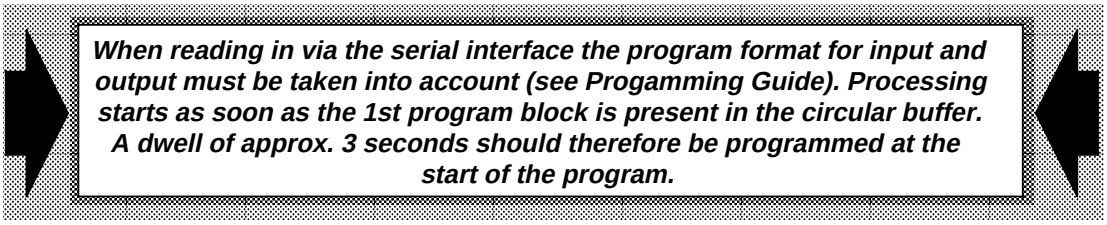
Any comments still contained in the circular buffer are displayed on the CRT in the alarm line. No comments will appear when processing is extremely fast.

When the EXECUTION FROM EXTERNAL function has been selected the following points should be observed:

- A program having the same number as the one to be read in is deleted.
- Simulation cannot be carried out.
- Programs cannot be edited, copied, shifted or renamed.
- Block search is only possible with calculation (START 1).
- Blocks in the part program containing errors can be displayed but not corrected.
- Only forward jumps are possible in the part program.
- If the part program uses subroutines, these must be contained in the part program memory.
- An error message appears whenever the interface via which a part program is to be read in is occupied.
- The EXECUTION FROM EXTERNAL function is retained in the event of power failure (if the unit is switched off, for example). Program parts still contained in the memory can be deleted by softkey operation or via ”Start program”.



The interface for the EXECUTION FROM EXTERNAL function is determined channel-specifically via a machine data.



When reading in via the serial interface the program format for input and output must be taken into account (see Progamming Guide). Processing starts as soon as the 1st program block is present in the circular buffer. A dwell of approx. 3 seconds should therefore be programmed at the start of the program.

© Siemens AG 1991 All Rights Reserved

3.3.3.3 OVERSTORE

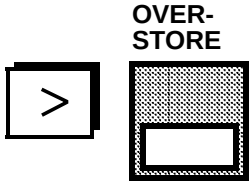
Using OVERSTORE, you can determine the spindle speed as well as the T, S, H and M functions. You can use this function to modify one or several values in the buffer memory. The program must be stopped for the "OVERSTORE" function.

Sequence of operation

A program is being processed in the **AUTOMATIC** operating mode.



When you press this key, the running program is stopped.



((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-HC01-0BA0
Seite 3–118 einmontieren))

You can now modify the following data using the correction pointer, numeric keys and input key:

- Tool number T . . .
- Tool offset number D . . .
- Spindle speed S . . .
- Auxiliary function H . . .
- Miscellaneous function M . . .



The program now operates with these new values until a new value appears in the program for the overstored function, or until you enter new values using OVERSTORE.

© Siemens AG 1991 All Rights Reserved

3.3.3.4 BLOCK SEARCH

For re-entering an interrupted program, there are three possibilities:

- Block search with calculation
- Block search without calculation
- Automatic block search with calculation from the last main block

A block search is likewise possible in a nest of subroutines. For this purpose, enter in the main program the number of the block in which the subroutine block is programmed. After this, enter the number, the number of passes and the block number of the subroutine.

The axes are not traversed during the search.

The output of the auxiliary functions during "Block search" is set as required as a machine data.

Depending upon the setting made during installation, the H-, M-, S- and T- functions are output either completely or in part, or fully suppressed.

Sequence of operation



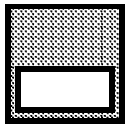


(AUTOMATIC)

The AUTOMATIC basic display appears on the CRT. The control is in the RESET state.

BLOCK SEARCH





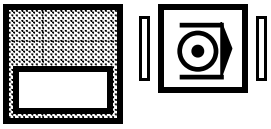
((Foto aus der Druckschrift SINUMERIK 880, Bedienungsanleitung, Bestellnr. 6ZB5 410-HC01-0BA0 Seite 3–119 einmontieren))

© Siemens AG 1991 All Rights Reserved

Using the numeric keys and input key enter the program number and number of the block sought.

The block search pointer does not differentiate between "N" (subsidiary block) and ":"(main block). The block is merely entered as a number.

START 1



Block search is carried out by the control and the program sequence activated.

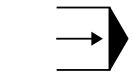
After BLOCK SEARCH, you can traverse the difference between the actual position and the calculated set position using the axis selector and direction keys in **REPOS** operating mode. Proceed as follows:



REPOS

The REPOS offset related to the end point of the preceding block appears on the display.

Using the axis selector switch on the machine control panel select the axis to be traversed and then press one of the direction keys. The control then brings the tool to the end point of the preceding block using linear interpolation.



AUTOMATIC



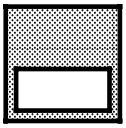
Program processing then continues.

Block search is activated after inputting the program number and number of the block sought by pressing the "START 2" softkey.

The block sought (main block or subsidiary block) must be in the main program plane. Once it is found, it is adopted as the current block.

© Siemens AG 1991 All Rights Reserved

START 2

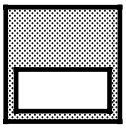


The block search is started.



After block search, further processing can be instigated by means of "NC Start" (program start) beginning at the block found.

START 3



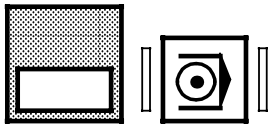
Automatic block search is activated using the "START 3" softkey.

This function contains block search without calculation to the last main block before the block sought and from there a block search with calculation to the block sought.

In this instance, a difference must be made between two possibilities:

- The block sought is a main block:
  It must be in the main program plane.
- The block sought is a subsidiary block:
  It may be in the main program plane or in one of the subroutine planes.

START 3



The automatic block search is carried out by the control and the program sequence activated.

Note:

- During block search, no message is output at the CRT.
- During automatic block search, "Block search active" is output at the CRT in the block search section.
- After a block search with calculation or automatic block search you can initiate no further block search with calculation, block search or automatic block search.
- This function has no effect if the target block is a block with G4 dwell time.
- After block search without calculation, it is possible to continue with block search with calculation or automatic block search, REPOS or NC start (program start).
- Block search with calculation is also possible in simulation.
- "N" may be input in the sought block number of the main program plane but not ":".
- "Block search with calculation" and "Automatic block search" are not permissible while the "TRANSMIT" function is active.

© Siemens AG 1991 All Rights Reserved

3.3.3.5 Interrupting the program

You can interrupt program processing with "Feed hold".

Sequence of operation

“Feed hold“:





The red "Feed hold" LED lights up.
The feed drives are brought to a standstill following the programmed paths.



When you press the "Feed start" key, the feed is re-enabled and the "Feed hold" LED goes out.

Note:

The "Feed hold" LED is also lit when a traverse inhibit prevents the start, e. g.:

- with failures of the control loop enable
- with signals from the monitoring function
- with 0 % Feedrate override **and** enable present

© Siemens AG 1991 All Rights Reserved

3.3.4 Manual data input-Automatic (MDI AUTOMATIC) mode

In this operating mode, you can process individual operations, block by block under NC control. After the blocks entered have been processed, they are deleted automatically.

Sequence of operation

Set the operating mode selector switch on the machine control panel to this symbol:



MDI AUTOMATIC

((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-HC01-0BA0
Seite 3–123 einmontieren))

Using the keyboard enter a block following the instructions in the Programming Guide and terminate each entry – including "L_F" (end of block) – using the input key.



The entered block is now processed by the control and subsequently deleted.

Note:

Before "Program start" is operated, you can enter several blocks (max. 256 characters).

The "Retentive (modal) input data" (e.g. feedrate) are retained.

They are deleted or reset:

- by changing the operating mode
- by RESET

Programming M2, M30 or M17 does not constitute a RESET or home position.

The current channel is specified on input of a character. Further inputs and NC start are then possible only in this channel. The channel can only be changed if all the inputs in the basic display have been deleted. This can be done by:

- Clearing the blocks after ”Program start”
- Block-by-block deletion using the keyboard
- Mode group RESET in the mode group assigned to the current channel

© Siemens AG 1991 All Rights Reserved

3.3.5 TEACH IN

Using the TEACH IN function in the **AUTOMATIC** operating mode you can generate a part program on a block-by-block basis.

You can test the program immediately.

Sequence of operation

Set the operating mode selector switch on the machine control panel to this symbol:

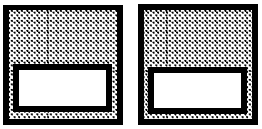


AUTOMATIC

((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-HC01-0BA0
Seite 3–124 einmontieren))

Enter the part program or subroutine number you require using the numeric keys and the input key.

PART PROGR. **TEACH IN PL. BACK**



The part program number selected appears on the display with the entry pointer below it ▮ .

Now enter a block (or several blocks) following the instructions in the Programming Guide and conclude each entry – including "L_F" (end of block) – using the input key.

You can also work with Operator prompting.
The part program is entered directly into the part program memory.



The blocks entered are now processed by the control.

You can delete the blocks not required, by entering the block no. "N ..." and operating the delete key.

Blocks traversed correctly remain – as entered – in the memory. They do not have to be acknowledged.

© Siemens AG 1991 All Rights Reserved

3.3.6 PLAYBACK

Using the PLAYBACK function after TEACH IN, you can program **linear movements**.

The position values are approached at jog traverse rate.

Positions traversed to are transferred into the program memory.

The blocks entered are transferred into the program selected in the **AUTOMATIC** basic display.

TEACH IN and PLAYBACK can be used alternately for generating a part program. When changing from AUTOMATIC to JOG or INC ... mode the TEACH IN and PLAYBACK functions are retained. After switching back from JOG or INC... to AUTOMATIC, TEACH IN/PLAYBACK must be reselected!

The ”NC Start” key must be pressed before the STORE softkey is operated the first time.

Sequence of operation

At least one complete part program block must be entered and traversed.

Set the mode selector switch on the machine control panel to one of the following symbols:



JOG

or

1, 10, 100, 1 000, 10 000

INC FEED ...

((Foto aus der Druckschrift SINUMERIK 880, Bedienungsanleitung, Bestellnr. 6ZB5 410-HC01-0BA0 Seite 3–125 einmontieren))

© Siemens AG 1991 All Rights Reserved

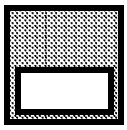
Using the axis selector switch on the machine control panel select the axis to be traversed and then move the tool to the required position with the direction keys.

Using the keyboard enter a block number "N..." and store it with the input key.

STORE

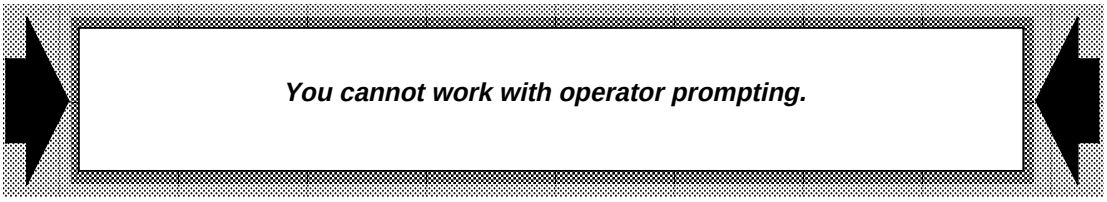


Axis assignment
(e.g. “X“)



The control transfers the value under **"Actual position"** for the axis selected into the selected block "N...".

After transferring the current positions for all axes, enter the required technology values, e.g. feedrate "F..." etc., and conclude each input – including "L_F" (block end) – with the input key.





This transfers the input block to the program memory.

Blocks not required can be deleted by entering the block no. "N...", and operating the delete key.

Blocks traversed correctly remain - as entered - in the memory. They do not have to be acknowledged.



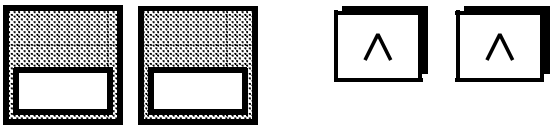
After each RESET using this key, execution of the program can be repeated.

© Siemens AG 1991 All Rights Reserved

Continuing TEACH IN/PLAYBACK after RESET

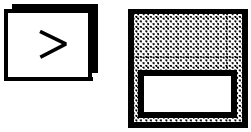
Set the mode selector switch to the symbol for **AUTOMATIC**.
Enter the desired part program number "%..." in the **AUTOMATIC** basic display and store it with the input key.

PART PROGR. **TEACH IN PL. BACK**



The AUTOMATIC basic display appears on the CRT.

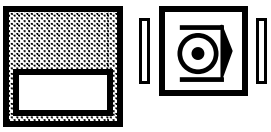
BLOCK SEARCH



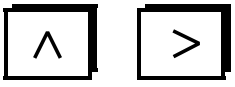
((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-HC01-0BA0
Seite 3–127 einmontieren))

Then enter the block no. "N..." of the last block in the part program and store it by pressing the input key.

START

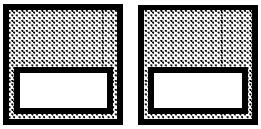


Block search is carried out by the control and the program sequence activated.



You return to the **AUTOMATIC** basic display .

PART PROGR. **TEACH IN PL. BACK**



The part program number selected appears on the display with the entry pointer below it ▮ .

Continue as described above.

© Siemens AG 1991 All Rights Reserved

3.3.7 JOG mode

In this mode, operation is NC-controlled in JOG*/*manual mode and not in a program.

You **traverse** the axes with the **direction keys**.

The traversing rate is preset as one machine data.

Sequence of operation

Set the operating mode selector switch on the machine control panel to this symbol:



JOG

((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-HC01-0BA0
Seite 3–128 einmontieren))

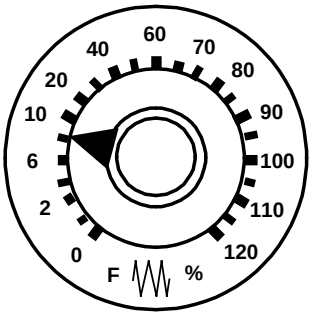
Using the axis selector switch on the machine control panel select the axis to be traversed and then position the tool using the direction keys.

The red "Feed hold" LED display must not be lit!
You can simultaneously traverse as many axes as channels are defined in the mode group.



You can select rapid traverse overlay using this key on the machine control panel.

© Siemens AG 1991 All Rights Reserved



You can modify the rapid traverse rate using the feedrate override switch.

The ”0 %” setting generally causes feed and rapid traverse to stop.

The traversing rate is preset as one machine data.

The feedrate can also be active for rapid traverse in the 0 % to 100 % range!

The spindle speed and T-, D-, S-, H-, and M-functions can be determined using **OVERSTORE** (see Section 3.3.3.2).

3.3.8 INC FEED 1 ... INC FEED 10 000 mode (Incremental)

In this mode, operation is NC-controlled in **"**JOG***/***manual**"** mode and not in a program; with each operation of a direction key, you traverse the axis concerned in the selected direction by the set increment. The increment values are settable as follows:

1, 10, **100**, **1 000**, **10 000** increments of display resolution.

The feedrate is defined as one machine data!

© Siemens AG 1991 All Rights Reserved

Sequence of operation

Set the operating mode selector switch on the machine control panel to the INC FEED symbol (5th to 9th switch position):

1, 10, 100, 1 000 or 10 000

((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-HC01-0BA0
Seite 3–130 einmontieren))

Using the axis selector switch on the machine control panel select the axis to be traversed and then position the tool using the direction keys.

The red "Feed hold" LED display must not be lit!

The direction keys can have different characteristics:

- ”Self retentive (modal)"
- "Jog operation"

With **"Self retentive"** the axis is always traversed by one increment, depending on the setting (1, 10, 100, 1000, 10000 increments of display resolution), when the key is pressed (irrespective of whether the key is pressed for a short or long time).

In **"Jog operation"** the axis is traversed for as long as the key remains pressed.

When the key is no longer being pressed, the traversing movement is stopped – even if the set increment has not yet been reached.

A machine data defines whether "Incremental" is traversed in "Self retentive" or "Jog operation" mode.

© Siemens AG 1991 All Rights Reserved

3.3.9 DRF

The DRF (differential resolver function) softkey function makes it possible to generate an additional, incremental zero offset using the "Handwheel".
(Value=1 increment of display resolution)

This zero offset is **not** taken into account for the actual position display.

The DRF function must be preselected in the master channel

(1st channel of a mode group).

Sequence of operation

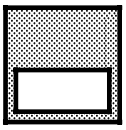
Set the operating mode selector switch to this symbol:



AUTOMATIC

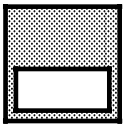
Now operate the following keys:

PROGRAM CONTROL



((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-HC01-0BA0
Seite 3–131 einmontieren))

DRF YES-NO



You activate the DRF offset via the handwheel by switching from "NO" to "YES".

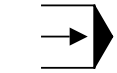
The "DRF handwheel" enable ("YES") is displayed on the CRT.

When you now operate the handwheel, you can effect the "DRF offset".

Note: The ”DRF” function cannot be used for axes for which the input resolution is less than the position control resolution nor for axes declared rotary axes.

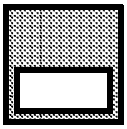
© Siemens AG 1991 All Rights Reserved

To display the value of the ”DRF offset” you set the mode selector switch to any operating mode (except PRESET), e. g. AUTOMATIC:



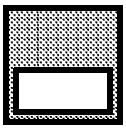
AUTOMATIC

SETTING DATA

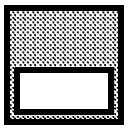




AXIAL



DRF OFFSET



((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-HC01-0BA0
Seite 3–132 einmontieren))

© Siemens AG 1991 All Rights Reserved

3.3.10 PRESET mode (Set actual value)

- In the **PRESET** operating mode, you can offset the control zero point to any point within the machine coordinate system.

  You enter the value for the offset into the actual value memory (preset).

  The actual value memory for all available axes can be preset.

  This preset results in a **PRESET** offset which is displayed on the CRT.

- If required, a tool compensation can be calculated into the **PRESET** offset.

  You enter the tool compensation data (compensation number, compensation direction and type) **before** ”Set actual value”.

  The value then entered is transferred into the actual value memory taking the tool compensation into account.

- **No movement** of the axes takes place with ”Set actual value”.
- The **PRESET** offset **remains stored**:
  – after "End of Program"
  – after "RESET"

One machine data defines whether the PRESET offset is automatically deleted:
- ***on switching on the control***
- ***on approaching reference point***

Sequence of operation

Set the operating mode selector switch to the following symbol:

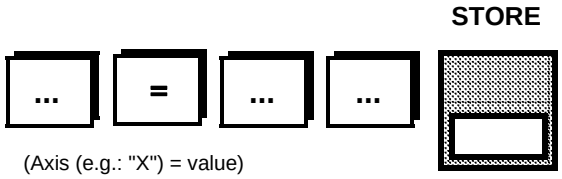


PRESET

((Mitgeliefertes Foto
Nr. 3–132 einmontieren))

© Siemens AG 1991 All Rights Reserved

1. **PRESET** offset **without** inclusive calculation of the tool compensation

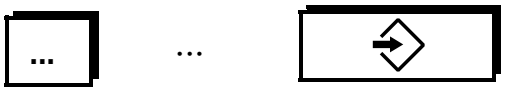


Using the keyboard, enter numeric "0" into the "Compensation number " input field, and store it using the soft-key ”STORE”.

The new "actual position" and the **PRESET** offset for the selected axis are displayed.

In the same way you enter the values for the remaining axes.

2. **PRESET** offset **with** inclusive calculation of the tool compensation

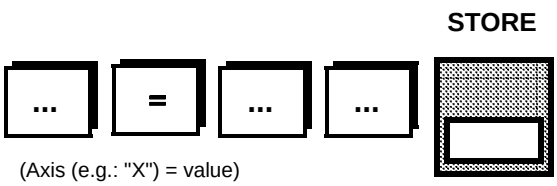


Using the keyboard, enter the number of the desired tool offset number D ... , into the ”compensation number” input field, and store it using the input key.



Using the keyboard, enter the identification for the X axis, into the ”Ident no.” input field. Store it using the input key.

Possible Ident numbers are:
2 allowance for L1
3 allowance for L2
4 allowance for radius**/**diameter



In the same way enter the identifier for all the axes in the "Identifier" input field (see Section 3.2.4). This defines the tool offset parameter (axis-specific assignment of the tool data).

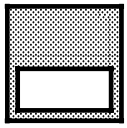
The new "actual position" and the **PRESET** offset for the selected axis are displayed.

The displayed value comprises: the entered value plus the calculated tool compensation.

© Siemens AG 1991 All Rights Reserved

Enter the values for all the other axes.

CLEAR
PRE-OFF



With this softkey you delete the **PRESET** offsets of the operating mode group selected.

3.3.11 REPOS mode (Repositioning)

After a program interruption – e.g. after switching from **AUTOMATIC** mode to **JOG** or "Incremental" (**INC...**) – you can traverse away from the contour.

AUTOMATIC operation is **not** aborted, i.e. the control is **not** brought to the reset state by a self-generated Reset:

Don't forget:

As described in Section 2.3.4, no Reset is generated by switching from AUTOMATIC mode to:

- ***JOG or***
- ***INC . . . or***
- ***REPOS***

When switching to modes other than those mentioned above, the control automatically generates a RESET; this brings the control into the reset state.

When traversing away from the contour, the distance moved is registered by the control. The distance to the interruption point is stored and displayed as the **REPOS** offset.

In **REPOS** mode, you can now approach to the interruption point using the direction keys along any path.

The direction key for the opposite direction is inhibited, and overtravel past the start position is not possible.

© Siemens AG 1991 All Rights Reserved

After a tool change, you can only use the "REPOS" mode with the same tool dimensions as before.
Otherwise you should use the following methods:

- ***"Block search" (see Section 3.3.3.3)***
- ***"Scratch" (see Section 3.3.12)***

Sequence of operation

Prerequisite: You have interrupted the program – e.g. by operating the "Feed hold" key – and subsequently have traversed away from the contour (see Section 3.3.7 for sequence of operation), in the **JOG** operating mode.

Set the operating mode selector switch on the machine control panel to this symbol:



REPOS

((Foto aus der Druckschrift
SINUMERIK 880,
Bedienungsanleitung,
Bestellnr. 6ZB5 410-HC01-0BA0
Seite 3–136 einmontieren))

The red "Feed hold" LED display is extinguished. Now select the axis to be traversed with the axis selector switch on the machine control panel and traverse to the point of interruption. When the point of interruption is reached the **REPOS** offset displayed is 0 and the direction keys are inhibited.

You can simultaneously traverse as many axes as channels are defined in the mode group.
The feedrate override switch is active and the rapid traverse override key is inactive.

© Siemens AG 1991 All Rights Reserved

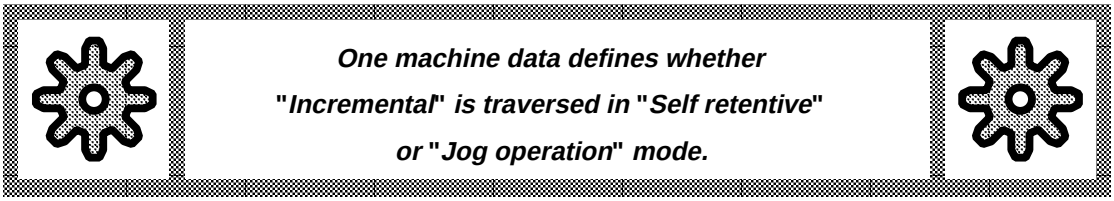
Note:

The direction keys can have two different characteristics:

- "Modal"
- "Jog operation"

With **"Modal"** the axis is always traversed by one increment, depending on the setting (1, 10, 100, 1000, 10000 increments of display resolution), when the key is pressed (irrespective of whether the key is pressed for a short or long time).

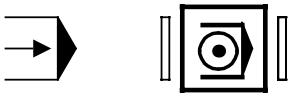
In **"Jog operation"** the axis is traversed for as long as the key remains pressed. When the key is no longer being pressed, the traversing movement is stopped – even if the set increment has not been reached.





You can interrupt the retraction of the axes with "Feed hold".

Set the operating mode selector switch on the machine control panel to **AUTOMATIC** mode when the contour has been reached:



AUTOMATIC

The program is resumed.

© Siemens AG 1991 All Rights Reserved

3.3.12 Scratch

If a tool breaks during program processing, you must leave the contour by changing to the **JOG** or **INC1...10000** operating mode in order to change the tool.

After entry of the new tool length compensation (the tool nose radius remains unchanged) you traverse the new tool to any point on the contour within the interrupted block (”Scratch”).

Sequence of operation

Prerequisite: A tool change has been carried out; the control is in **JOG** mode.

Using the direction keys, now traverse to any point **within the interrupted block**. Select the axis to be traversed with the axis selector switch on the external machine control panel and then operate one of the direction keys.

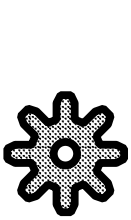
Set the operating mode selector switch on the machine control panel to **AUTOMATIC** mode again:



AUTOMATIC



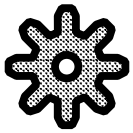
The end point of the block is approached and program processing resumed.



In a block with circular interpolation (G02, G03) scratch must take place within a very narrow range.

This range is fixed with machine data. (Standard definition via MD 9).

If this range is exceeded Alarm 3018 ”Excessive distance to contour” is output.



© Siemens AG 1991 All Rights Reserved

© Siemens AG 1991 All Rights Reserved

4 Alarms and Notes for the Operator

4.1 Alarms

4.1.1 General

The control contains constantly active monitoring functions which can recognize faults in the NC, the interface control and the machine early enough for damage to the workpiece, tool or machine to be largely eliminated. If a fault does occur, first of all machining is interrupted, then the drives are brought to a standstill and the reason for the fault is stored and displayed as an alarm. At the same time the PLC is informed that an NC alarm exists.

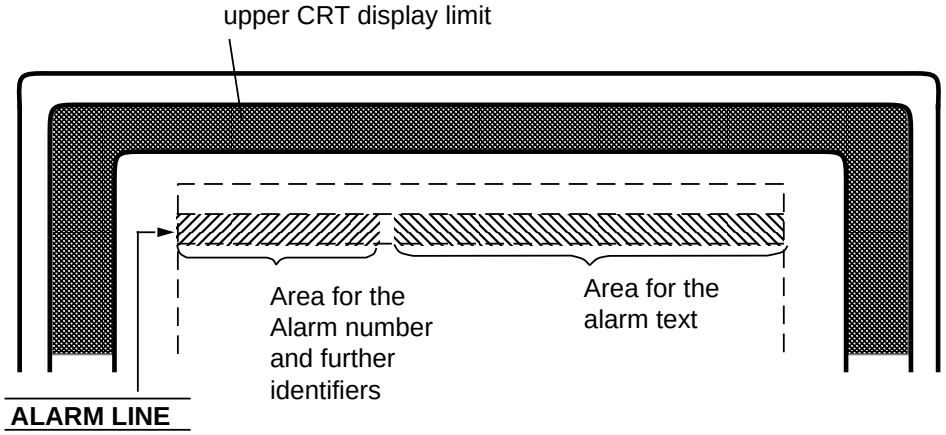
Monitoring functions exist for the following areas:

- Read-in
- Format
- Measuring circuit cables
- Encoders and drive
- Contour
- Spindle speed
- Enable signals
- Voltage
- Temperature
- Microprocessors
- Serial interfaces
- Transfer between NC, COM and PLC
- Voltage level of the backup battery
- System program memory
- User program memory

4.1.2 Display of alarms on the CRT

Messages from the monitoring system are displayed in the alarm line of the CRT display.

The alarm line is the second display line from the top.

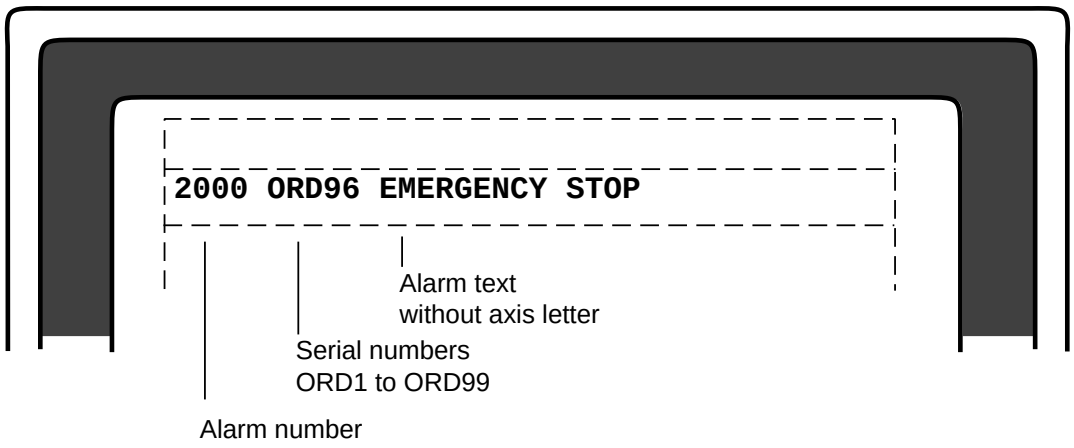


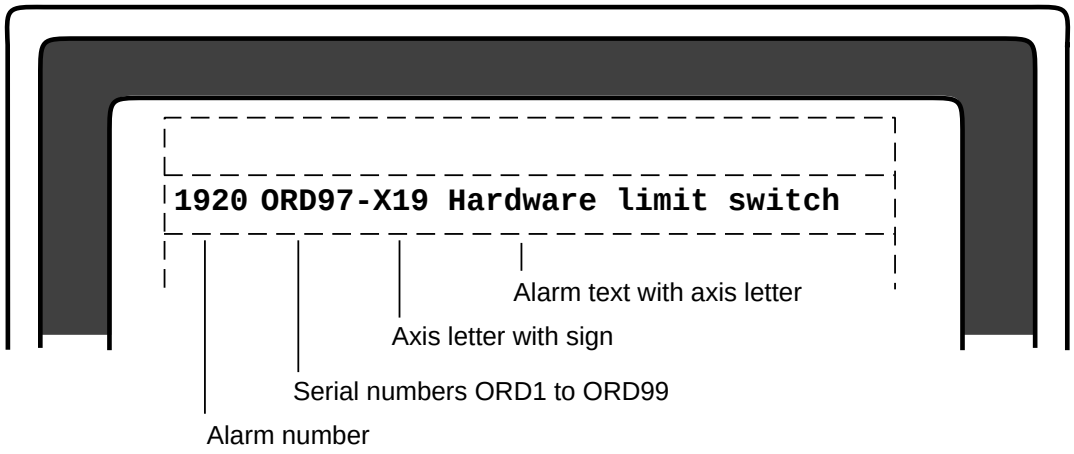
© Siemens AG 1991 All Rights Reserved

4.1.3 Display representation

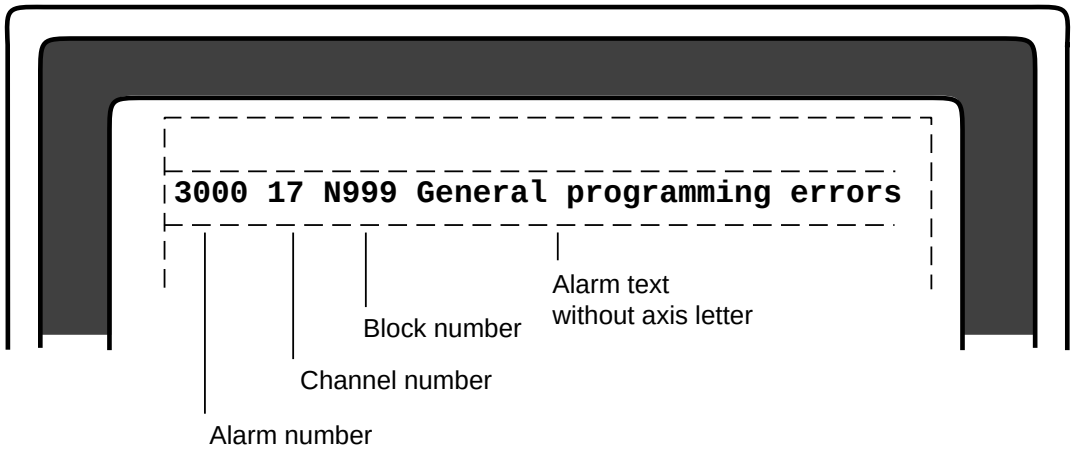
There are 3 types of display representation for alarms:

1. Alarms with serial number

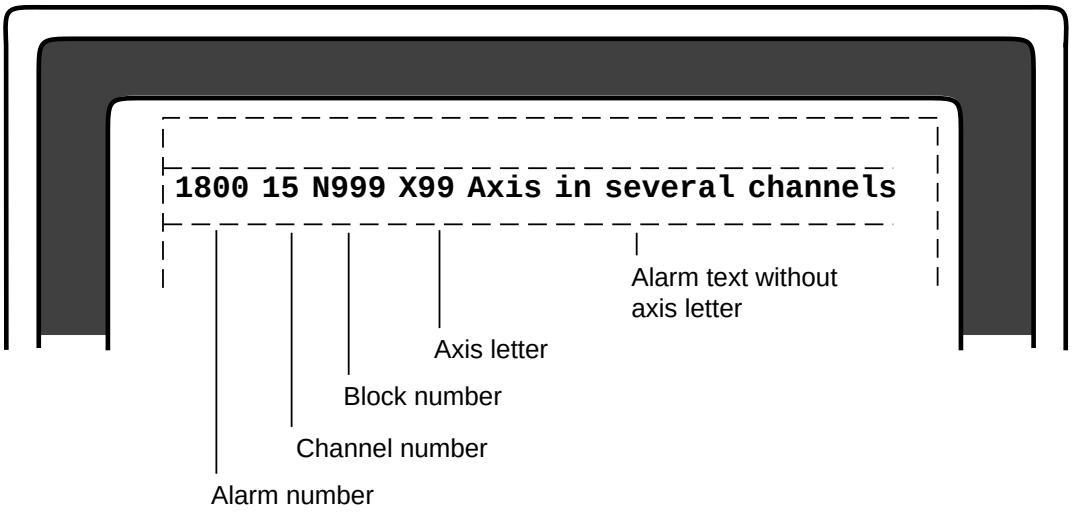




2. Alarms with block number



© Siemens AG 1991 All Rights Reserved



Note:

Axis- and spindle-specific alarms in this Operator's Guide are marked with an asterisk in the last decimal place. Example: Alarm number 156*

The asterisk stands for the relevant axis or spindle. The following assignments apply:

Spindles	**Axes**
1 = 1st spindle	0 = 1st axis
2 = 2nd spindle	1 = 2nd axis
3 = 3rd spindle	2 = 3rd axis
4 = 4th spindle	3 = 4th axis
5 = 5th spindle	4 = 5th axis
6 = 6th spindle	.
	.
	.
	23 = 24th axis

Examples for spindles: Alarm 225*[c)] spindle speed too high

2251	Alarm refers to the 1st spindle
2252	Alarm refers to the 2nd spindle
2256	Alarm refers to the 6th spindle

Examples for axes: Alarm 156* setpoint speed too high

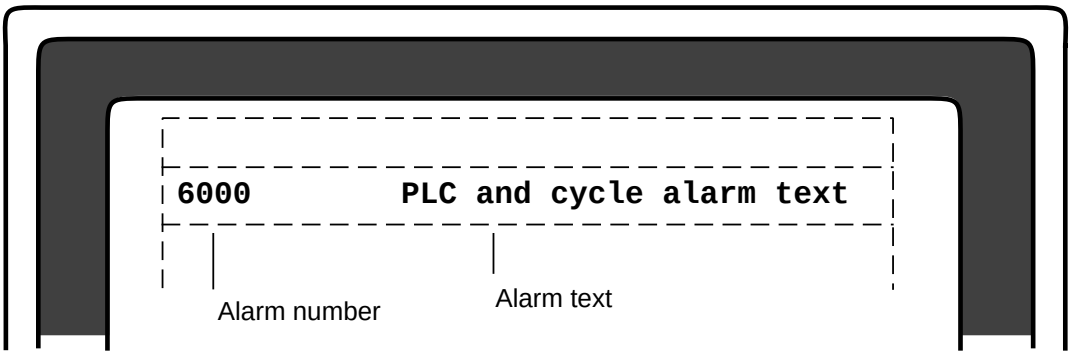
1560	Alarm refers to the 1st axis
1565	Alarm refers to the 6th axis
1570	Alarm refers to the 11th axis
1583	Alarm refers to the 24th axis

---

c) *up to and including Software Version 6*

© Siemens AG 1991 All Rights Reserved

3. PLC and cycle alarms



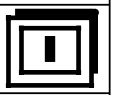
Note:

PLC and cycle alarms are determined by the machine tool manufacturer or the user himself. They are therefore not included in the alarm description.

© Siemens AG 1991 All Rights Reserved

4.1.4 Alarm numbers and groups/clearing alarms

The alarms are divided into different groups. The table below shows the relationship between alarm number, alarm group and alarm clearing function.

Alarm number	Alarm group	Alarm cleared by ...
2...6 8...15 40...89	POWER ON alarms	Restart the control 
16...35	RS232C (V.24) alarms	1. Call the softkey menu including the "DATA IN-OUT function" [1] 2. Operate "DATA IN-OUT" softkey 3. Operate "STOP" softkey 
100*[2]... 120* 140*...196* 1004*...1008* 1016* 1024*...1036* 1052*...1060* 1204*...1216*	RESET alarms/ axis-specific	Operate RESET key 
225* [c] 228*...231* [c] 2000*...2031* [d]	RESET alarms/ spindle-specific	Operate RESET key 
128*... 136* 226*... 227* 1000* 1012* 1040* 1200*	POWER ON alarms/ axis-specific/spindle-specific	Restart control 
2000 ... 2195 2410 ... 2427	RESET alarms/general	Operate RESET key 
1 and 7 3000 ... 3246	Acknowledge alarms	Operate acknowledge key 
1044* [d]	Acknowledge alarms axis-specific	Operate acknowledge key 
4100...4200	Cycle alarms	Operate acknowledge key 
5000-5099	User cycle alarms	Operate acknowledge key 
6000...9999	PLC error message	Operate acknowledge key 
6000...9999	PLC operatial message	These messages are reset automatically by the PLC program.

Tabular overview with assignment of alarm no. and clear function

---

1) Explanatory note: The DATA IN-OUT function can be called in the following operating modes: AUTOMATIC/JOG/REFPOINT/INCFEED 1....10000/PRESET

2) The asterisk (*) stands for an axis/spindle number.

c) up to and including Software Version 6

d) from GA2, Software Version 1

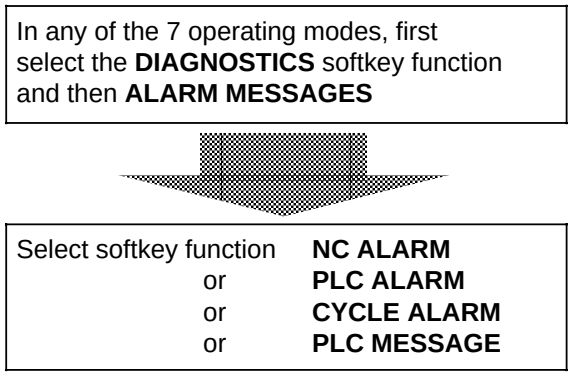
© Siemens AG 1991 All Rights Reserved

4.1.5 DIAGNOSTICS/selection of further alarms

When the diagnostics react, the reason could be **several different faults occurring at the same time**.

Only the alarm with the **lowest alarm number** is displayed in the alarm line.

If you need an overview of any other existing alarms/messsages, proceed as follows:



4.1.6 Effects of alarms

In the section ”List of alarms/alarm description” the four types of effect are listed. The detailed meaning of the effects is explained below.

- **Interruption of processing**

  The program is interrupted and can be resumed at the interrupt point after the fault has been remedied.

  With channel-specific alarms, processing is interrupted only in the channel concerned.

- **NC START inhibit**

  NC START is not enabled when an alarm is active.

  The current processing is not stopped. However, if processing is interrupted (e. g. by NC STOP or Single block) it cannot be restarted until the cause of a fault has been remedied and the alarm cleared.

  With channel-specific alarms, NC START is inhibited only for the channel concerned.

- **Cancellation of Mode Group Ready**

  The MODE GROUP READY signal is cancelled for the master channel. As a result, maximum braking current is applied to the axes or spindles of the mode group for rapid deceleration (Rapid stop).

- **Opening the NC ready relais**

  The output on the dual/multiport is disabled. The ready state of the control can only be reestablished via POWER ON.

© Siemens AG 1991 All Rights Reserved

4.1.7 Listing of alarms/alarm description

The following list is based on the presumption that servicing is not carried out by the operator.

1	**Battery alarm-power supply** 
Scan:	• On POWER ON • Cyclic
Effect:	• None
Explanation:	• The battery voltage has dropped so far that backup (of the program memory, for example) cannot be guaranteed if the control is switched off
Remedy:	Replace the battery next to the power pack under power and acknowledge the alarm with the ”Acknowledge key” after making spot checks in the part program memory for variable data
Caution:	The control must not be switched off until the battery has been replaced, otherwise stored data will be lost

2	**Overtemperature** 
Scan:	Cyclic
Effect:	• Output of temperature error at PLC user interface flag 24.2
Explanation:	The fan exhaust temperature is higher than 55 °C (permissible range 0 °C to 55 °C)
Remedy:	Check temperature in the cabinet

© Siemens AG 1991 All Rights Reserved

4	Incorrect unit system	
Scan:	• On POWER ON • After modification of NC MD	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay	
Explanation:	• Different unit systems have been used for MD 18000 display resolution and MD 502 input resolution. Both data must have the same unit system. • The alarm is displayed with the serial number	
Remedy:	Check and correct the machine data combination. Then delete the alarm via POWER ON.	

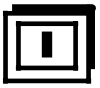
5	Incorrect MIB parameter	
Scan:	On formatting the user program memory; ”UM FORMAT” softkey	
Effect:	Only the MIB parameters 0 to 199 are defined	
Explanation:	• The alarm is displayed with the serial number. • The R and MIB parameters take up so much space in the user memory that the user program memory has become smaller than 16Kbytes • MD 4 < MD 5 overlapping of both MIB areas • MD 4 < 200 standard MIB area exceeded • MD 5 > 500 MIB buffer area exceeded	
Remedy:	Correct NC MD 4 or NC MD 5 and reformat the user program memory	

6[c)]	Battery alarm machine data	
Scan:	• On POWER ON • Cyclic	
Effect:	None	
Explanation:	• The alarm is displayed with the serial number. • The battery voltage has dropped so far that backup of the NC MDs cannot be guaranteed	
Remedy:	Replace the NC MD card with a new one and re-enter the NC MDs and PLC MDs (from the beginning of 1991, MD backup in central battery)	
Caution:	The control must not be switched off before the replacement has been made, otherwise stored data will be lost	

c) up to and including Software Version 6

© Siemens AG 1991 All Rights Reserved

7	**Battery alarm operator panel**	
Scan:	• On POWER ON • Cyclic	
Effect:	None	
Explanation:	• The battery voltage has dropped so far that backup of the setup data for the Terminal Emulator cannot be guaranteed	
Remedy:	Replace battery in the operator panel under power and acknowledge the alarm with the CANCEL key after the setup data have been checked and corrected if necessary	
Caution:	The control must not be switched off until the battery has been replaced, otherwise stored data will be lost	

8	**Wrong axis/spindle assignment**	
Scan:	• After modification of NC MD • On POWER ON	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay	
Explanation:	The NC machine data for the axis assignment (MD 200*) and the spindle assignment (MD 400*) have been input incorrectly or have been reversed. Error in MD 461* C axis definition	
Remedy:	Check and correct axis and spindle assignment machine data. Delete alarm via POWER ON.	

9	**Memory too small for UMS**	
Scan:	On POWER ON	
Effect:	Illegal UMS	
Explanation:	• Excessive modification of the standard area, RAM area available insufficient	
Remedy:	Avoid excessive modification of the standard area	

© Siemens AG 1991 All Rights Reserved

10	UMS error	
Scan:	On POWER ON	
Effect:	NC START disabled	
Explanation:	• The alarm is displayed with the serial number • UMS selected but no UMS inserted or EPROMS on UMS are empty. • UMS RAM not yet loaded	
Remedy:	• Insert correct UMS • Check NC MD 5015 bit 6	

11	Wrong UMS identifier	
Scan:	On POWER ON	
Effect:	NC START disabled	
Explanation:	• The alarm is displayed with the serial number • UMS RAM not yet loaded • Wrong EPROM inserted • Linking error (operating system)	
Remedy:	• Load UMS RAM • Insert correct EPROM • Relink UMS	

12	Part program memory wrongly formatted	
Scan:	Cyclic	
Effect:	• Processing interruption • NC START disabled	
Explanation:	• The alarm is displayed with the serial number • A fault has been ascertained on the memory module or a hardware repair has been made and the part program memory not yet reformatted	
Remedy:	Reformat memory (CLEAR PART PROGRAM softkey) or replace memory	

© Siemens AG 1991 All Rights Reserved

13	**Check NC MD**	
Scan:	On POWER ON	
Effect:	• The alarm is displayed with the block number • Opening of NC Ready Relay	
Explanation:	Validity check during ramp-up showed inconsistency.	
Remedy:	Check MD in "Block number" field. Check comments for each machine data concerned.	

14	**Check PLC MD**	
Scan:	During POWER ON	
Effect:	• The alarm is displayed with the block number • Opening of NC Ready Relay	
Explanation:	Validity check during power up showed inconsistency.	
Remedy:	Check MD in "Block number" field. Check comments for each machine data concerned.	

16	**Parity error RS232C (V.24)**	
Scan:	On reading data into the RS232C (V.24) interface	
Effect:	• RS232C (V.24) transmission interrupted • Last block declared invalid	
Explanation:	• Alarm is displayed with the block number. • The alarm can only arise if the setting data (SD) ”with parity bit” applies. The character commenced (8 data and 1 parity bit) has the wrong parity. This alarm has nothing to do with character parity errors RS232C (V.24) in ISO or EIA punched tapes (Alarm 23)	
Remedy:	• Check SDs 5011, 5013, 5018, 5020 • Test external device	

© Siemens AG 1991 All Rights Reserved

17	**Overflow error RS232C (V.24)**	Soft-key
Scan:	On reading data into the RS232C (V.24) interface	
Effect:	• RS232C (V.24) transmission interrupted • Last block declared invalid	
Explanation:	• Alarm is displayed with the block number • The external device has sent a new character although the NC has not yet processed the old one	
Remedy:	• Check SDs 5011, 5013, 5018, 5020 • Test external device	

18	**Frame error RS232C (V.24)**	Soft-key
Scan:	On reading data into the RS232C (V.24) interface	
Effect:	• RS232C (V.24) transmission interrupted • Last block declared invalid	
Explanation:	• The alarm is displayed with the block number • Number of stop bits is incorrect • Baud rate is incorrect	
Remedy:	• Check SDs 5011, 5013, 5018, 5020 • Test external device	

19	**I/O device not ready RS232C (V.24)**	Soft-key
Scan:	On request of data from the NC via the RS232C (V.24) interface	
Effect:	No files are read in	
Explanation:	• Alarm is displayed with the block number • The DSR signal from the external device is Low	
Remedy:	• Start external device • Do not use DSR	

© Siemens AG 1991 All Rights Reserved

22	Time watch-dog RS232C (V.24)	Soft-key
Scan:	Following an NC request to output or read in data; after a delay of 60 seconds	
Effect:	No data are received or output	
Explanation:	• Alarm is displayed with the block number • The NC cannot output a signal for 60 seconds – External device blocks the CTS signal (**c**lear **t**o **s**end) for longer than 60 seconds – External device does not send DC1 within 60 s when control signals (DC1 to DC4) are used • The NC has not received a signal for 60 seconds	
Remedy:	• Check external device and switch on • Check cable and plug in • Switch off time watch-dog (SD bit)	

23	Character parity error RS232C (V.24)	Soft-key
Scan:	On reading in data via the RS232C (V.24) interface of the NC	
Effect:	• RS232C (V.24) transfer interrupted • The last block declared invalid	
Explanation:	• Alarm is displayed with the block number • Depending on the definition of the start of program ”%” or ”EOB”, the NC automatically determines the ISO or EIA code on receipt of this character and thus the character parity On checking the following characters, it has been established that one character did not have the set parity	
Remedy:	Check punched tape and data format output of the data input device. The incorrect block number is displayed.	

© Siemens AG 1991 All Rights Reserved

24	**Invalid EIA character RS232C (V.24)**	Soft-key
Scan:	On reading in data via the RS232C (V.24) interface of the NC	
Effect:	• RS232C (V.24) transfer interrupted • The last block declared invalid	
Explanation:	• Alarm is displayed with the block number • An EIA character with the correct parity was read in, but the character was **not** defined in EIA code	
Remedy:	• Check punched tape • Check SD (EIA code for ”@”) • Check SD (EIA code for ”:”) • Check SD (EIA code for ”=”) • Check SD (EIA code for ”]”) • Check SD (EIA code for ”[”) • Check SD (EIA code for ”,”)	

26	**Block > 120 characters RS232C (V.24)**	Soft-key
Scan:	On reading in data via the RS232C (V.24) interface of the NC	
Effect:	• RS232C (V.24) transfer interrupted • The last block declared invalid	
Explanation:	• The alarm is displayed with the block number. • The part program block that has been read in contains more than 120 characters. Only the actually stored characters are counted (no spaces, no CR, ...)	
Remedy:	Divide block into 2 or more blocks	

27	**Data input disabled RS232C (V.24)**	Soft-key
Scan:	On reading in data via the RS232C (V.24) interface of the NC	
Effect:	No data have been read in	
Explanation:	• Alarm is displayed with reference to block • An attempt has been made to read in NC MDs in normal mode • An attempt has been made to transfer UMS data to the NC although the UMS has not been enabled or not inserted • An attempt has been made to read in a subroutine which already exists as a cycle in the EPROM • The interface signal "Cycle inhibit" is active.	
Remedy:	• Read in NC MDs in installation mode or enter password • Insert UMS module and fit with UMS submodules or enable UMS via machine data • Reset interface signal "Cycle inhibit" (DB 48).	

© Siemens AG 1991 All Rights Reserved

28	Circ. buffer overflow RS232C (V.24)	Soft-key
Scan:	On reading in data into the NC via the RS232C (V.24) interface	
Effect:	• RS232C (V.24) transfer interrupted • The last blocks declared invalid	
Explanation:	• Alarm is displayed with the block number • Data transfer speed so high that more characters are transferred than can be processed by the NC • RTS signal has no effect on the input device (RTS causes the input device to STOP) • Transmission speed (baud rate) too high	
Remedy:	• Use RS232C (V.24) transmission instead of 20 mA • Use 3rd serial interface	

29	Block > 254 charac. RS232C (V.24)	Soft-key
Scan:	On reading in tool data into the NC via the RS232C (V.24) interface	
Effect:	• RS232C (V.24) transfer interrupted • The last block declared invalid	
Explanation:	• Alarm is displayed with the block number • The block that has been read in has more than 254 characters. All characters read in are counted (including blanks, CR, LF ...)	
Remedy:	Divide block into 2 or more blocks	

30	Part program memory overflow RS232C (V.24)	Soft-key
Scan:	On reading in programs via the RS232C (V.24) interface of the NC	
Effect:	• RS232C (V.24) transfer interrupted • The last block declared invalid	
Explanation:	• Alarm is displayed with the block number • The maximum part program memory space is full	
Remedy:	Clear old programs to provide memory space for reading in new programs. The block number containing an error is displayed.	

© Siemens AG 1991 All Rights Reserved

31	**Program memory full RS232C (V.24)**	Soft-key
Scan:	On reading in programs via the RS232C (V.24) interface of the NC	
Effect:	No data have been read in	
Explanation:	• Alarm is displayed with the block number • The available part program memory is full.	
Remedy:	• Read out and clear old part programs that are not needed anymore to provide memory space for new part programs • Use larger part program memory if necessary	

32	**Data format error RS232C (V.24)**	Soft-key
Scan:	On reading in data via the RS232C (V.24) interface of the NC	
Effect:	• RS232C (V.24) transfer interrupted • The last block declared invalid	
Explanation:	• Alarm is displayed with the block number • The permissible number of decades after an address is not correct • The decimal point occurs in the wrong place • The part programs or subroutines are not defined or concluded correctly (check heading!)	
Remedy:	Check the program to be read in (incorrect block number is displayed)	

33	**Different programs same number RS232C (V.24)**	Soft-key
Scan:	On reading in programs via the RS232C (V.24) interface of the NC	
Effect:	No data have been entered/stored	
Explanation:	• Alarm is displayed with the block number • If a new program is read in which has the same program number as the one already stored in the NC, both programs are compared and, because they are different, the NC outputs Alarm 33. The new program is **not stored.**	
Remedy:	• Delete old program (DELETE) • Rename old program (RENAME)	

© Siemens AG 1991 All Rights Reserved

34	Operator error RS232C (V.24)	Soft-key
Scan:	On starting another RS232C (V.24) interface when the other one is already active	
Effect:	No data have been read in	
Explanation:	• Alarm is displayed with the block number • The RS232C (V.24) transmission is started in the NC and the PLC sends DATA START once more to the NC	
Remedy:	Restart RS232C (V.24) transmission	

35	Reader error RS232C (V.24)	Soft-key
Scan:	Only if setting data are set for the Siemens reader	
Effect:	• RS232C (V.24) transfer interrupted • The last block declared invalid	
Explanation:	• Alarm is displayed with the block number • The Siemens reader has output a group error message	
Remedy:	• Restart RS232C (V.24) data transfer • If the error recurs: replace Siemens reader	

40	COM CPU fault	
Scan:	Cyclic	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay	
Explanation:	• The alarm is displayed with the block number. • Hardware or software fault in the COM CPU area • No PLC-CPU fault reported • An incorrect or faulty EPROM submodule has been inserted.	
Remedy:	Replace hardware and/or software	

© Siemens AG 1991 All Rights Reserved

41	NC CPU fault	
Scan:	Cyclic	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay	
Explanation:	• The alarm is displayed with the block number. • Hardware or software fault in the range of the respective NC CPU • An incorrect or faulty EPROM submodule has been inserted.	
Remedy:	• On installation: check jumper on the bus board • Replace hardware and/or software	

42	SERVO CPU fault	
Scan:	Cyclic	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay	
Explanation:	• The alarm is displayed with the block number. • Hardware or software fault in the range of the respective SERVO CPU • Processor overloaded • Measuring circuit module fault • An incorrect or faulty EPROM submodule has been inserted.	
Remedy:	• Check NC MD 156 or 160 to 163 • Reduce the number of axes/spindles on the SERVO CPU • Deselect tool nose radius compensation or assign axis to a different SERVO CPU • Replace hardware and/or software	

© Siemens AG 1991 All Rights Reserved

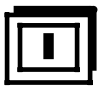
43	**PLC-CPU fault**	
Scan:	Cyclic	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay	
Explanation:	• Alarm is displayed with the block number • Hardware or software error in the PLC or NC COM area • PLC in STOP status • NC does not assume Reset status and continues displaying the status of the instant of interruption • PLC machine data error • Error in the PLC user program • An incorrect or faulty EPROM submodule has been inserted.	
Remedy:	• Read out I-STACK • Establish cause of error using the error list in the installation lists • Replace hardware and/or software	

44	**Part program memory not available**	
Scan:	On POWER ON	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay	
Explanation:	• The alarm is displayed with the serial number. • Part program memory missing or not recognised or does not reach the minimum size	
Remedy:	• Check specified memory size (machine data) • Clear memory and format	

46	**Incorrect TO parameter display**	
Scan:	Cyclic	
Effect:	none	
Explanation:	• The alarm is displayed with the serial number. • NC MD 13 too large	
Remedy:	Modify NC MD	

© Siemens AG 1991 All Rights Reserved

47	Illegal TO assignment	
Scan:	Cyclic	
Effect:	None	
Explanation:	• The alarm is displayed with the serial number. • Value of NC MD 210 is greater than 16 • TO starting numbers in NC MDs 211 to 226 have not been entered in ascending order • Input value in NC MDs 1040 to 1047 is greater than the number of TO ranges or 0 is defaulted in the TO range	
Remedy:	Check NC MDs	

57d)	SERVO timeout	
Scan:	On POWER ON	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay	
Explanation:	• The alarm is displayed with the serial number • The set position control time for axes and spindle is too short	
Remedy:	Resynchronize (switch on/off), alter position control time	

58d)	Parameterization error	
Scan:	On POWER ON	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay Enter setpoint 0 and follow-up mode	
Explanation:	Measuring circuit assigned more than once (see alarm 1000* and 2016*)	
Remedy:	Correct machine data 200* and MD400*.	

d) from GA2, Software Version 1

© Siemens AG 1991 All Rights Reserved

59[d]	**SIMODRIVE FAILURE**	
Scan:	Cyclic	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay Enter setpoint 0 and follow-up mode	
Explanation:	• The cable to the processor module has been pulled out or the processor module has been switched off during operation	
Remedy:	Resynchronization in cyclic operation is necessary (switch off/on).	

60[d]	**Drive PM without voltage**	
Scan:	Cyclic	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay Enter setpoint 0 and follow-up mode	
Explanation:	SINUMERIK switched on before processor module.	
Remedy:	This alarm is automatically deleted when the processor module is switched on.	

61[d]	**Drive not synchronized**	
Scan:	On POWER ON	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay Enter setpoint 0 and follow-up mode	
Explanation:	Processor module has received negative acknowledgement, triggered either by a hardware fault or incorrect jumpering.	

d) from GA2, Software Version 1

© Siemens AG 1991 All Rights Reserved

63	**Connection fault** 
Scan:	Cyclic
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay
Explanation:	• The alarm is displayed with the serial number • The transmission connection between the COM CPU and the operator panel is interrupted. Normal operation is no longer possible.
Remedy:	• Check the fiber-optic or copper cable between the operator panel and the central controller • Check COM or operator panel CPU

64	**Incorrect checksum** 
Scan:	On POWER-ON
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay
Explanation:	• Alarm is displayed with the block number • An EPROM error has been found on switching on the control: – Checksum of one card is wrong. The card number is displayed as block number. This number corresponds to the number of the servo CPU with the faulty EPROM. The block number 0 corresponds to the NC CPU (error LED flashes). – Checksum of Restart EPROM wrong
Remedy:	• Check card slots and correct if necessary • Check EPROMS

65c)	**Wrong slot** 
Scan:	On POWER ON
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay
Explanation:	• Alarm is displayed with the block number • On switching on the control it has been found that one module has been inserted in the wrong bus slot
Remedy:	Check module slots and replace modules if necessary

c) up to and including Software Version 6; from Software Version 1 onwards all the system software is stored on one memory submodule.

© Siemens AG 1991 All Rights Reserved

66[c)]	**Wrong software system**	
Scan:	On POWER ON	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay	
Explanation:	• Alarm is displayed with the block number • On switching on the control it has been found that the wrong software system has been inserted in the control e. g. 810 software system instead of 880 software system	
Remedy:	Insert correct software system	

67	**1st computer link not ready**	
Scan:	Cyclic or after POWER ON	
Effect:	Message frame interchange between the host computer and the NC is not possible.	
Explanation:	• The alarm is displayed with the serial number • Host computer and NC are not synchronous caused by a wrong specification or a fault on the interface module. Message frame interchange is not possible.	
Remedy:	• Check configuration of interface module • Check machine data specified for the computer link • Check whether host computer is ready or has been connected	

68	**2nd computer link not ready**	
Scan:	Cyclic or after POWER ON	
Effect:	Message frame interchange between the host computer and the NC is not possible.	
Explanation:	• The alarm is displayed with the serial number • Host computer and NC are not synchronous caused by a wrong specification or a fault on the interface module. Message frame interchange is not possible.	
Remedy:	• Check configuration of interface module • Check machine data specified for the computer link • Check whether host computer is ready or has been connected	

c) *up to and including Software Version 6; from Software Version 1 onwards all the system software is stored on one memory submodule.*

© Siemens AG 1991 All Rights Reserved

69	Servo CPU not available	
Scan:	On POWER ON	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay	
Explanation:	• Alarm is displayed with the block number • The servo circuit of an axis/spindle has been defined in MD 200* or MD 400* on a non-existing servo CPU.	
Remedy:	Correct MD 200* or MD 400*	

70	Fault in at least one channel	
Scan:	On POWER ON	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay	
Explanation:	• The alarm is displayed with the serial number • A wrong assignment in machine data has been chosen on installation. The NC is not operational without channel assignment. At least 1 channel must be assigned to each NC CPU.	
Remedy:	Check channel assignment machine data and correct (MD 100*)	

71	NC CPU not available	
Scan:	On POWER ON	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay	
Explanation:	• Alarm is displayed with the block number • A wrong assignment in machine data has been chosen on installation. One channel has been assigned to a non-existing NC CPU.	
Remedy:	Check channel assignment machine data and correct (MD 102*)	

© Siemens AG 1991 All Rights Reserved

72	One channel too many in mode group	
Scan:	On POWER ON	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay	
Explanation:	• Alarm is displayed with the block number • A wrong assignment in machine data has been chosen on installation. Too many channels have been assigned to one mode group.	
Remedy:	Check mode group assignment machine data and correct (MD 100*)	

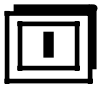
74	One channel too many in the NC CPU	
Scan:	On POWER ON	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay	
Explanation:	• Alarm is displayed with the block number • A wrong assignment in machine data has been chosen on installation. Too many channels have been assigned to one NC CPU.	
Remedy:	Check channel assignment machine data and correct (MD 102*)	

75	Too many servos for CPU	
Scan:	On POWER ON	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay	
Explanation:	• Alarm is displayed with the block number • Too many real axes and spindles have been defined in the machine data.	
Remedy:	Check machine data and correct	

© Siemens AG 1991 All Rights Reserved

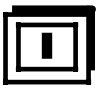
76	**Wrong measuring circuit assignment**	
Scan:	On POWER ON	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay	
Explanation:	• Alarm is displayed with the block number • A wrong assignment in machine data has been chosen on installation. One axis/spindle has been assigned to a wrong measuring circuit or a wrong measuring circuit module.	
Remedy:	Check machine data and correct	

77	**Illegal mode group no. of axis**	
Scan:	On POWER ON	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay	
Explanation:	• Alarm is displayed with the block number • The alarm is only generated when the mode group number = 0 or > 8.	
Remedy:	Check machine data 360* and correct	

78	**Illegal mode group no. of spindle**	
Scan:	On POWER ON	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay	
Explanation:	• Alarm is displayed with the block number • Spindle has been assigned to a wrong mode group	
Remedy:	Check machine data 453* and correct	

© Siemens AG 1991 All Rights Reserved

79	**Illegal mode group no. of channel**	
Scan:	On POWER ON	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay	
Explanation:	• Alarm is displayed with the block number • Channel has been assigned to a wrong mode group	
Remedy:	Check machine data 100* and correct	

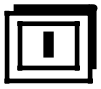
80	**Illegal NC no. of channel** c)	
Scan:	On POWER ON	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay	
Explanation:	• Alarm is displayed with the block number • Channel has been assigned to a wrong NC CPU	
Remedy:	Check machine data 102* and correct	

81	**More than two 2D interpolations**	
Scan:	On each block change	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay	
Explanation:	• Alarm is displayed with the block number • The NC is an export version. In the NC, 2 axes (2D interpolation) have been programmed in more than 2 channels in one program block.	
Remedy:	• Modify program • Do not run more than 2 programs simultaneously	

c) up to and including Software Version 6

© Siemens AG 1991 All Rights Reserved

82	Wrong simulation channel selected	
Scan:	On pressing the UM FORMAT softkey	
Effect:	None	
Explanation:	• The alarm is displayed with the serial number • Graphics simulation is not possible because a wrong value has been specified in NC MD 173	
Remedy:	Correct NC MD 173 (input value 1 or 2)	

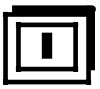
83	Wrong no. of SWD parameters	
Scan:	On pressing the UM FORMAT softkey	
Effect:	None	
Explanation:	• The alarm is displayed with the serial number • Graphics simulation is not possible because a wrong value has been specified in NC MD 172	
Remedy:	Correct NC MD 172 (only input value 8 permissible)	

84	Incorrect coupled axis grouping	
Scan:	• On POWER ON • On warm restart	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay	
Explanation:	• The alarm is displayed with the serial number • An incorrect coupled motion grouping has been specified in NC MD 876 to 899 or 5156 to 5182, e. g. – Axes do not belong to the same mode group – Axes have different position control resolutions – Different axis type (linear axis/rotary axis) – Axes are not available (NC MD 564* bit 7=0) – Axes are fictitious axes – Leading axis=following axis	
Remedy:	• Correct NC MDs 876 to 899 • Correct NC MDs 5156 to 5182	

© Siemens AG 1991 All Rights Reserved

85	**Wrong coupled motion combination**	
Scan:	• On POWER ON • On warm restart	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay	
Explanation:	• The alarm is displayed with the serial number • The undefinable bit combination ”01” has been entered in the NC MDs 5156 to 5182	
Remedy:	Correct NC MDs 5156 to 5182	

87	**Illegal software limit switch**	
Scan:	On modification of MD	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay	
Explanation:	• Alarm is displayed with reference to block • An excessive value has been entered in NC MDs 224*to 236*. The maximum traversing range of the individual axes is determined by the axis-specific position control resolution set and the input resolution.	
Remedy:	Correct NC MD 224* to 236* if necessary	

88	**Interpolation greater than 3D[c)]4D[d)]**	
Scan:	On processing part programs in AUTOMATIC or MDA mode	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay	
Explanation:	• Alarm is displayed with reference to block • More than 4 axes are programmed in one block in the NC part program block or the 5D function is not provided	
Remedy:	• Check part program • Divide program block into two separate blocks • Have function option checked by a service expert and retrofit if necessary	

c) up to and including Software Version 6

d) from GA2, Software Version 1

© Siemens AG 1991 All Rights Reserved

89	**More than two 3D interpolations** 
Scan:	On processing part program blocks in AUTOMATIC or MDA mode
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay
Explanation:	• Alarm is displayed with reference to block • In the NC, more than 3 axes are programmed in one program block in more than 2 channels
Remedy:	• Modify program • Process up to 2 programs simulaneously

100*	**Illegal leadscrew error compensation grid spacing** 
Scan:	Cyclic
Effect:	• NC START disabled • Cancellation of Mode Group Ready
Explanation:	• The alarm is displayed with the serial number • The permissble range (1 to 1000 lead screw error compensation points) has been exceeded in axis traversing movements. • Lead screw error compensation for rotary axes. A wrong value for the relevant axis has been entered in NC machine data 324*. Formula: 360/input value = number of grid spacings The number of grid points must be an interger! Examples: NC MD 324*=10 360/10=360000 grid points NC MD 324*=12 360/12=300000 grid points NC MD 324*=14 360/14=257143 grid points The value 14 is not permissible because the result must always be an interger
Remedy:	Modify NC MD 324*

104*	**DAC limit[c]** **Rpm setpoint warning limit[d]** 
Scan:	Cyclic
Effect:	• NC START disabled • The drive will oscillate if too high a velocity is entered • Alarm 156* ”Setpoint speed too high” is triggered if there is a fault in the drive
Explanation:	• The alarm is displayed with the serial number • The DAC setpoint is higher than entered in NC MD 268* (maximum DAC setpoint). It is not possible to increase the DAC setpoint further.
Remedy:	• Traverse more slowly • Check actual value (encoder) • Check NC MD 268* • Check the drive actuator

c) up to and including Software Version 6
d) from GA2, Software Version 1

© Siemens AG 1991 All Rights Reserved

108*c)	**Overflow with variable increment weighting**	
Scan:	On each axis movement	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • The servo enable is cancelled after the time stored in NC MD 156 has elapsed (servo enable relay drops out) • Follow-up mode	
Explanation:	• The alarm is displayed with the serial number • If a value is entered in NC MDs 364*, 368* (pulse weighting) which is not 1:1, the actual part value must be multiplied by the control. With rapid traverse of the axis, the register will overflow in the case of a fault. The reference point is lost.	
Remedy:	• Reduce maximum velocity (NC MD 280*); maximum velocity depends on NC MDs 364* and 368* • Check MDs 364* and 368* • Check actual values (encoder)	

c) up to and including Software Version 6

© Siemens AG 1991 All Rights Reserved

112*	**Zero speed control**	
Scan:	• On standstill • On clamping • On deceleration	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay • The servo enable is cancelled after the time stored in NC MD 156 has elapsed (servo enable relay drops out) • Follow-up mode	
Explanation:	• The alarm is displayed with the serial number • The following error could not be cleared faster than the time entered in NC MD 156 • On clamping, the limit defined in NC MD 212* was exceeded • A mechanically clamped axis has been pushed out of position • Fault in the control device (actuator), at the tacho-generator, at the motor, in the NC measuring circuit hardware or at/on the encoder • Wrong position feedback polarity direction on installation	
Remedy:	• NC MD 212* (clamping tolerance) must be greater than NC MD 204* (exact stop limit) • NC MD 156 (servo enable cutoff delay) must be large enough to allow the following error to be cleared within this period (this only applies if NC MD 372* = 0) • NC MD 372* (zero-speed control delay) must be large enough to allow the following error of the axis to be cleared within the period entered • Check actual values (encoder) and position control polarity • Reduce acceleration (accelaration/delay so large that the circuit voltage in the drive will reach the permissible limit)	

© Siemens AG 1991 All Rights Reserved

116*	**Contour monitoring**	
Scan:	When machining in AUTOMATIC/MDA, but **not :** • When accelerating • When decelerating • At speeds lower than that in NC MD 336* (contour speed)	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • The servo enable is removed after the time stored in NC MD 156 has elapsed (servo enable relay drops out) • Follow-up mode	
Explanation:	• The alarm is displayed with the serial number • With a speed greater than that in NC MD 336* the tolerance band in NC MD 332* has been exceeded • When accelerating or decelerating the axis has not reached the new speed within the time defined by the servo gain factor	
Remedy:	• Increase the tolerance band in NC MD 332* • Check the servo gain factor • Check the optimization of the speed controller • Check the actual values (encoder) • Check the free movement of the axes • Reduce acceleration	

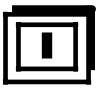
© Siemens AG 1991 All Rights Reserved

120*c)	**Illegal axis specification**	
Scan:	• On standstill • On clamping • When decelerating	
Effect:	• Processing interruption • NC START disabled • Servo enable is cancelled after the time in NC MD 156 has elapsed (servo enable relay drops out) • Follow-up mode	
Explanation:	• The alarm is displayed with the serial number • Missing setting via MD200* or MD384* of the axis concerned Example: MD2000=01020101 and MD3840=00000000 • The module number set in MD200* or MD 384* is larger than the number of measuring circuit modules available Example: MD2000=04010000 and only 3 measuring-circuit modules inserted • The connection number set in MD200* or MD384* is larger than the number of available connections on the module in question Example: MD3840=02070000, the 2nd measuring-circuit module is an SPC module and has only 6 terminals • Connection number for an input is assigned to an output and vice versa Example: MD3840=01030000, the 1st measuring-circuit module is an HMS module on which terminal no. 3 is an input terminal • Input and output assignment is not compatible with the insertion sub-module Example: MD2000=01040101, the 1st measuring-circuit module is an HMS module with output submodule servo command 6FX1132-5BAxx in its submodule slot 1	
Remedy:	• Check and correct MD 200* and 384* of the axis concerned. These machine data must both be specified or be zero. They must also agree with the hardware configuration.	

c) up to and including Software Version 6

© Siemens AG 1991 All Rights Reserved

128*	**Measuring circuit not available**	
Scan:	On POWER ON	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready	
Explanation:	• The alarm is displayed with the serial number • In NC MD 200* or 400* measuring circuits have been assigned to axes or spindles which have been recognised by the NC as non-existent after POWER-ON RESET • Measuring circuit module removed or defective	
Remedy:	• Check machine data • Check measuring circuit module	

132*	**Control loop hardware**	
Scan:	Cyclic	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • The servo enable is removed after the time stored in NC MD 156 has elapsed (servo enable relay drops out) • Follow-up mode	
Explanation:	• The alarm is displayed with the serial number • The measuring circuit differential signals – are not in phase – have a short-circuit to earth – are missing • In the case of distance coded reference marks, reference point approach has not taken place between 2 basic distances of the reference marks.	
Remedy:	• Check that the measuring circuit connector is inserted • By plugging in the measuring circuit short-circuit connector it can be checked whether the measuring circuit module is in working order • Check the differential signals with an oscilloscope • Replace the measuring sensors • Check NC MD 200*, 384* • Check IAR MD 100*[c)]	
Caution:	The alarm can only be deleted with POWER ON	

c) up to and including Software Version 6

© Siemens AG 1991 All Rights Reserved

136*	**Measuring system dirty** 
Scan:	Cyclic
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready
Explanation:	• The alarm is displayed with the serial number • In measuring systems with contamination signal (e.g. EXE), the measuring system signals a fault to the NC
Remedy:	Check the measuring system following the manufacturer's instructions
Caution:	The alarm can only be cancelled with POWER ON

140*	**Encoder monitoring** 
Scan:	On axis motions (setpoint not equal to 0) if the function has been activated via machine data bit
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready
Explanation:	• The alarm is displayed with the serial number • Only check 32 bit servo CPUs • Too many directional changes have been recognized in the actual position value between position control timers. Possible causes are: – One encoder track is broken or connected to a fixed potential (typical fault) – The encoder is precisely located on an edge and the hardware tends to oscillate (effect of contamination) – Fault in the electronics leads to clattering of one track (effect of contamination)
Remedy:	Check encoder track

144*	**Zero mark monitoring** 
Scan:	Cyclic, depending on specification of tolerance band for differential pulses
Effect:	• Processing interruption • NC START disabled
Explanation:	• The alarm is displayed with the serial number • Pulses have been lost as a result of errors in transfer or other faults beyond the permissible tolerance band with reference to one encoder revolution. The tolerance band is checked by a reference meter.
Remedy:	• Check encoder pulses and establish permissible deviation • Check transmission path • Switch off monitoring temporarily by specifying a value greater than 16 in the machine data (NC MD 216*)

© Siemens AG 1991 All Rights Reserved

148* 152*	**Software limit switch plus** **Software limit switch minus**	
Scan:	At each axis movement	
Effect:	• Processing interruption • NC START disabled	
Explanation:	• The alarm is displayed with the serial number • The software limit switches are only active after reference point approach • Depending on the PLC interface signal 2ND SOFTWARE LIMIT SWITCH ACTIVE, the 1st or 2nd software limit switch has been approached • Traverse away from the limit switch in opposite direction	
Remedy:	• Check NC MDs 224*, 228*, 232*, 236*	

156*	**Set speed too high**[c)] **Rpm setpoint alarm limit**[d)]	
Scan:	Cyclic	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • The servo enable is removed after the time stored in NC MD 156 has elapsed (servo enable relay drops out) • Follow-up mode	
Explanation:	• The alarm is displayed with the serial number • A higher set speed has been output in the control than that specified in NC MD 264* • The motor cannot follow the speed command value • On installation: wrong position control direction • The following error is too large because either the acceleration rate is too high or because of incorrect encoder matching (feedback polarity, evaluation) • Fault in drive unit • Mechanical clamping	
Remedy:	• Check whether the value in NC MD 264* is higher than the value in NC MD 268* • Check the drive • Check the position feedback polarity • Check the set speed cable • Check the actual values (pulse encoder) • Check IAR MD 124* and 136*[c)]	

c) up to and including Software Version 6

d) from GA2, Software Version 1

© Siemens AG 1991 All Rights Reserved

160*	**Drift too high**	
Scan:	Only with semiautomatic drift compensation	
Effect:	• NC START disabled • The green "Position not yet reached" LED lights up • No traverse movement possible	
Explanation:	• The alarm is displayed with the serial number • The drift to be compensated by the NC itself has risen to over approx. 500mV	
Remedy:	• Execute drift compensation in NC MD 272* (see also Installation Guide) Operation: 1. Select NC MD 272* 2. Press the ”EDIT” key • Check that the drift has been correctly adjusted on the drive unit	

164*	**Synch. (coupled) axis programmed**	
Scan:	On processing in AUTOMATIC or MDA mode	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready	
Explanation:	• The alarm is displayed with the serial number • Despite the fact that coupled motion has been selected a following axis was programmed	
Remedy:	Program program axis	

168*	**Servo enable refused for traversing axis**	
Scan:	At each axis movement	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • The servo enable is removed after the time stored in NC MD 156 has elapsed (servo enable relay drops out) • Follow-up mode	
Explanation:	• The alarm is displayed with the serial number • The axis-specific servo enable signal was removed by the PLC during traversing	
Remedy:	Check the PLC program	

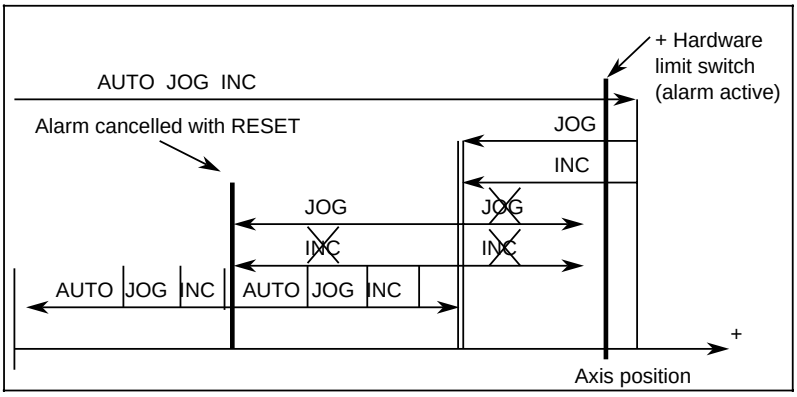
© Siemens AG 1991 All Rights Reserved

172* 176*	**+ Working area limitation - Working area limitation**	
Scan:	Cyclic after reference point approach	
Effect:	• Processing interruption • NC START disabled	
Explanation:	• The alarm is displayed with the serial number • The working area limitation in the setting data has been reached	
Remedy:	• Check the working area limitation in setting data (selection via the SETTING DATA and AXIAL DATA softkeys) • Check G25/G26 in the part program and modify if necessary • Move relevant axis clear of limitation • Cancel ”Reference point reached” signal (e. g. define axis as ”parking axis”)	

180*	**Axis activated in several channels**	
Scan:	On processing in AUTOMATIC or MDA mode	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready	
Explanation:	• This alarm is displayed with the block number • On simultaneous processing of two or more programs in different channels the same axis has been programmed in both programs (channels)	
Remedy:	• Check both programs • Insert L999 or @ 714 • Stop one channel with NC STOP	

© Siemens AG 1991 All Rights Reserved

188* 192*	**+ Hardware overtravel (limit) switch - Hardware overtravel (limit) switch**	
Scan:	Cyclic	
Effect:	• Processing interruption • NC START disabled • Direction key disabled in approach direction	
Explanation:	• The alarm is displayed with the serial number • Limit switch reached	
Remedy:	• Traverse in opposite direction (correct operating mode)	



• Check PLC user program
• Check LIMITATION signals in DB 32

196*	**Synch. (coupled) axis assigned 2 x**	
Scan:	On processing in AUTOMATIC or MDA mode	
Effect:	• NC START disabled • Processing interruption	
Explanation:	• The alarm is displayed with the serial number • Two leading axes have been programmed whose following axes are identical Axis X axis Y Axis Z axis Y • Two leading axes have been programmed, one of these also being the following axis of the other leading axis Axis X axis Y Axis Y axis Z	
Remedy:	Correct program	

© Siemens AG 1991 All Rights Reserved

2000	**Emergency Stop**	
Scan:	Cyclic	
Effect:	• Processing interruption • NC START disabled • Follow-up mode	
Explanation:	• The alarm is displayed with the serial number • The EMERGENCY STOP signal is output from the PLC to the NC	
Remedy:	• Check with PLC STATUS • Check whether the EMERGENCY STOP cam was traversed to, or whether the EMERGENCY STOP switch was operated • Check PLC user program	

2031	**Weighting factor too large/small**	
Scan:	Cyclic	
Effect:	• Processing interruption • NC START disabled	
Explanation:	• The alarm is displayed with the serial number • On account of the weighting factor the actual axis speed is so high that the maximum permissible speed would be exceeded with the axis-specific position control resolution set	
Remedy:	• Check NC MD 388* • Program lower speed • Reduce feedrate/rapid traverse override	

2032[d)]	**Error in thread**	
Scan:	• During NC START and after block search with calculation • When switching to C axis operation	
Effect:	• Processing interruption • NC START disabled	
Explanation:	• The alarm is displayed with the block number • The gear stage was incorrect when switching over to C axis operation • FOLLOW-UP MODE or PARKING AXIS was active when tapping for the C axis was selected • Selection is not possible from basic setting G205	
Remedy:	• Program gear change • Check channel number • Check PLC signals; check NC MD 521*.4 • Delete PARKING AXIS signal	

d) from GA2, Software Version 1

© Siemens AG 1991 All Rights Reserved

2035	Feedrate limitation	
Scan:	On processing in AUTOMATIC or MDA mode	
Effect:	None	
Explanation:	• The alarm is displayed with the serial number • The programmed speed is greater than the path speeds resulting from the maximum speeds of the axes • The programmed distance to go in the thread block is equal to zero	
Remedy:	• Do not program a spindle speed with G95/G33 • Check NC MD 280* • Check part program	

2036	G35 thread lead decrease error	
Scan:	On thread cutting	
Effect:	• Processing interruption • NC START disabled	
Explanation:	• Alarm is displayed with the block number • The pitch decrease in threading is so high that at the end of the thread, a diameter of 0 would arise	
Remedy:	Program a lower pitch decrease or a shorter thread	

2037	Prog. S value too high	
Scan:	On processing part programs in AUTOMATIC or MDA mode	
Effect:	None	
Explanation:	• The alarm is displayed with the serial number • The programmed spindle speed in AUTOMATIC is too high • An excessive feedrate has been calculated for thread cutting (G33/G34/G35/G37) as a result of the programmed spindle speed	
Remedy:	Program a lower spindle speed	

© Siemens AG 1991 All Rights Reserved

2038	Path feed too high	
Scan:	Cyclic	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready	
Explanation:	• Alarm is displayed with the block number • The axis speed has caused the programmed path feed to increase so much that the maximum permissible axis speed would be exceeded with the position control resolution set	
Remedy:	• Program a lower path feed • Check interpolation condition in the part program block	

2039	Reference point not reached	
Scan:	On starting a part program/program block in AUTOMATIC or MDA mode	
Effect:	None	
Explanation:	• The alarm is displayed with the serial number • The reference point has not been traversed to by at least one axis and NC START has been pressed in MDA or AUTOMATIC mode.	
Remedy:	• Approach reference point • The alarm does not appear if NC MD 5004 "NC START without reference point" is set. • This alarm can be disabled axis-specifically in NC MD 560*	

2041	Program not in memory	
Scan:	On specification of a program number and subsequent NC START	
Effect:	• Processing interruption • NC START disabled	
Explanation:	• Alarm is displayed with the axis and block number • The pre-selected program is not in the memory • A non-existing subroutine is called in the main program • The contour for the stock removing cycle does not exist	
Remedy:	• Look at directory • Check program	

© Siemens AG 1991 All Rights Reserved

2042	Parity error in memory	
Scan:	On processing in AUTOMATIC or MDA mode	
Effect:	• Processing interruption • NC START disabled	
Explanation:	• Alarm is displayed with the channel number and block number • One or more characters in the memory have been corrupted and can no longer be identified. These characters are output in the "Correction block" or in the "EDITOR" as "?".	
Remedy:	• Correct the program in "EDITOR" or delete the complete block and re-enter it if necessary	

2043	Prog. error transformation	
Scan:	On processing in AUTOMATIC or MDA mode	
Effect:	NC START disabled	
Explanation:	• Alarm is displayed with the channel number and block number • Programming of real axis with transformation selected • Programming of fictitious axes with transformation deselected • Selection of transformation although transformation has already been deselected • Programming of traversing movements in the selection block of transformation	
Remedy:	Correct program	

2044	External processing error	
Scan:	On starting an external program	
Effect:	• Processing interruption • NC START disabled	
Explanation:	• Alarm is displayed with the channel number and block number • The alarm is displayed: – if an external program selected is already being processed in another channel – if an interface (V.24 or File transfer) is to be started in several channels, or if the interface is already busy – if the 2nd interface of an interface pair (1st and 2nd RS232C (V.24), 3rd and 4th RS232C (V.24)) is to be started while the 1st interface is busy	
Remedy:	• Check NC machine data 130* • Check the active interfaces	

© Siemens AG 1991 All Rights Reserved

2045	Program already available	
Scan:	On NC START, if ”Processing an external program” has been selected	
Effect:	Processing not started	
Explanation:	• The alarm is displayed with the channel number and block number • A program having the same program number as the external program to be processed is already in the part program memory	
Remedy:	Rename the existing program or clear	

2046	Block > 120 characters	
Scan:	On processing in AUTOMATIC or MDA mode	
Effect:	• Processing interruption • NC START disabled	
Explanation:	• The alarm is displayed with the channel number and block number • An ”L_F” in the memory has been corrupted so that a block containing more than 120 characters exists	
Remedy:	Insert ”L_F” without deleting the complete block	

2047	Option not available	
Scan:	On processing in AUTOMATIC or MDA mode	
Effect:	• Processing interruption • NC START disabled	
Explanation:	• The alarm is displayed with the channel number and block number • A function has been programmed that is not included in the function set of the control • Setting data 204* "Thread start angle" has been edited although the option EXTENDED THREAD PACKAGE is not available	
Remedy:	• Correct program • Check NC MD • Have option checked by Service and retrofit if necessary	

2048	Circle endpoint error	
Scan:	On processing in AUTOMATIC or MDA mode	
Effect:	• Processing interruption • NC START disabled	
Explanation:	• The alarm is displayed with the channel number and block number • The programmed circle end point does not lie on the circle • The end point is out by more than the value entered in NC MD 7	
Remedy:	Correct program	

© Siemens AG 1991 All Rights Reserved

2056	Transf. centre travel	
Scan:	On processing in AUTOMATIC (Block search) or MDA mode	
Effect:	• Processing interruption • NC START disabled • Block search interrupted	
Explanation:	• The alarm is displayed with the channel number and the block number • A part program block has been programmed with selected transformation TRANSMIT that initiates a movement directly through the transformation centre • The path from the start to the end position of the block to be searched passes directly through the transformation centre	
Remedy:	• Correct program • On block search: modify start position in JOG	

2057	Thread/revolutional feedrate option	
Scan:	On processing a part program block when this option is selected	
Effect:	• Processing interruption • NC START disabled	
Explanation:	• The alarm is displayed with the channel number and the block number • A thread has been programmed using G33, G34, G35, G37 although this option is not available in the control • Revolutional feedrate G95 has been programmed • In the 880 T the "Revolutional feedrate" NC MD bit has not been set • A function has been programmed that is not included in the function set of the control	
Remedy:	• Correct program • Check NC MD • Have option checked by Service and retrofit if necessary	

2058	3D interpolation option not available	
Scan:	On processing program blocks in AUTOMATIC or MDA mode	
Effect:	• Processing interruption • NC START disabled	
Explanation:	• The alarm is displayed with the channel number and the block number • Three or more axes have been programmed in one block	
Remedy:	• Correct program • Check NC MD • Have option checked by Service and retrofit if necessary	

© Siemens AG 1991 All Rights Reserved

2059	**G92 program error**	
Scan:	On processing a relevant part program block	
Effect:	• Processing interruption • NC START disabled	
Explanation:	• The alarm is displayed with the channel number and the block number • An illegal address character has been used • The unit and working diameter factor is zero	
Remedy:	• Check program block • Check machine data	

2060	**TO, ZO program error**	
Scan:	On processing program blocks in AUTOMATIC or MDA mode	
Effect:	• Processing interruption • NC START disabled	
Explanation:	• The alarm is displayed with the channel number and the block number • Tool type is 0 (no tool) • A non-existent tool compensation number has been selected • The values in the selected zero offsets or tool compensations are too large	
Remedy:	• Check specifications in the program block • Check machine data specifications • Cancel alarm via the Reset key	

2061	**General program error**	
Scan:	On processing a part program or program block in AUTOMATIC or MDA mode	
Effect:	• Processing interruption • NC START disabled	
Explanation:	• The alarm is displayed with the channel number and the block number • The input resolution determines the range limitation for the positional data. If the programmed path length exceeds the permissible path distance to go, an alarm is activated.	
Remedy:	• Check the program section and correct the incorrect specification • Check the input resolution selected	

© Siemens AG 1991 All Rights Reserved

2062	Feed missing/not programmed	
Scan:	On processing program blocks in AUTOMATIC or MDA mode	
Effect:	• Processing interruption • NC START disabled	
Explanation:	• The alarm is displayed with the channel number and the block number • Revolutional feedrate G95 programmed greater than 50 mm/min • No revolutional feedrate programmed • No F value programmed • No feedrate for soft approach/exit • Zero is specified in MD 280* for the maximum speed of one axis	
Remedy:	• Check specifications in program block • Check machine data specifications • Cancel alarm via the Reset key	

2063	Thread lead (leadscrew) too high	
Scan:	On thread cutting with G33/G37	
Effect:	• Processing interruption • NC START disabled	
Explanation:	• The alarm is displayed with the channel number and the block number • The thread lead can be specified in the program under I, J, K. The programmed value exceeds the permissible value depending on the specified display resolution.	
Remedy:	• Correct program block whose block number and channel number is shown in the alarm display	

2064	Rounding wrongly programmed for rotary axis	
Scan:	On processing in AUTOMATIC or MDA mode	
Effect:	• Processing interruption • NC START disabled • The programmed movement in the block is **not** executed	
Explanation:	• The alarm is displayed with the channel number and the block number • If you round to either half or full degrees on a rotary axis, the control will monitor whether the programmed positions correlate with the rounding	
Remedy:	• Program the correct rotary axis position • Check MD 560* bits 2 and 3 • Check whether the interface signal CLEAR DISTANCE TO GO has been set (no automatic rounding!)	
Caution:	• In the JOG, INC modes, the control automatically rounds to valid values. In the AUTOMATIC or MDA modes, the control only monitors the programmed positions, without rounding itself. • The INC mode is not permissible	

© Siemens AG 1991 All Rights Reserved

2066	**Thread lead increase or decrease too large**	
Scan:	On processing in AUTOMATIC or MDA mode	
Effect:	• Processing interruption • NC START disabled	
Explanation:	• The alarm is displayed with the channel number and the block number • A thread lead or pitch increase or decrease of more than 16 mm/rev (0.6 inch/rev) has been programmed	
Remedy:	Program a smaller thread lead increase/decrease	

2067	**Max. axis speed = 0**	
Scan:	On processing in AUTOMATIC or MDA mode	
Effect:	• Processing interruption • NC START disabled	
Explanation:	• The alarm is displayed with the channel number and the block number • The resulting maximum speed of one axis is equal to 0	
Remedy:	Check NC MD 280*	

2069	**5D TLC (tool length correction) not possible**	
Scan:	On processing in AUTOMATIC mode	
Effect:	• Processing interruption • NC START disabled	
Explanation:	• The alarm is displayed with the channel number and the block number • Cutter radius selected • No linear interpolation selected • Option or NC MD not set or set incorrectly • ”3D interpolation” option not set • Export version	
Remedy:	• Check program • Check machine data	

2072	**Incorrect input value**	
Scan:	On processing in AUTOMATIC or MDA mode	
Effect:	• Processing interruption • NC START disabled	
Explanation:	• The alarm is displayed with the channel number and the block number • Programmed input for contour definition cannot be calculated	
Remedy:	• Check program • Check input values	

© Siemens AG 1991 All Rights Reserved

2073	**No intersection point**	
Scan:	On processing in AUTOMATIC mode	
Effect:	• Processing interruption • NC START disabled	
Explanation:	• The alarm is displayed with the channel number and the block number • Calculation of the contour definition gives no intersection point with the values programmed	
Remedy:	Check programmed specifications	

2074	**Incorrect angle value**	
Scan:	On processing in AUTOMATIC or MDA mode	
Effect:	• Processing interruption • NC START disabled	
Explanation:	• The alarm is displayed with the channel number and the block number • Angle 360° is programmed • The angle value has no meaning for the contour described	
Remedy:	Check programmed values and correct	

2075	**Incorrect radius value**	
Scan:	On processing in AUTOMATIC or MDA mode	
Effect:	• Processing interruption • NC START disabled	
Explanation:	• The alarm is displayed with the channel number and the block number • Radius value too large • Radius value not allowed for the contour described	
Remedy:	Check programmed values	

2076	**Incorrect G02/G03**	
Scan:	On processing in AUTOMATIC or MDA mode	
Effect:	• Processing interruption • NC START disabled	
Explanation:	• The alarm is displayed with the channel number and the block number • Direction of circle not possible for the described contour	
Remedy:	Correct program	

© Siemens AG 1991 All Rights Reserved

2077	**Incorrect block sequence**	
Scan:	On processing part programs in AUTOMATIC mode	
Effect:	• Processing interruption • NC START disabled	
Explanation:	• The alarm is displayed with the channel number and the block number • Several blocks are needed for calculation of contour definition – Block sequence not correct – Insufficient information (underdefined)	
Remedy:	Check programmed values	

2078	**Incorrect: input parameter**	
Scan:	On processing part programs in AUTOMATIC or MDA mode	
Effect:	• Processing interruption • NC START disabled	
Explanation:	• The alarm is displayed with the channel number and the block number • Programmed parameter sequence not allowed • Parameter sequence not complete for the described contour Example: `N10... X60 B15 LF` (Z axis missing) `N20... X90 B10 LF`	
Remedy:	Check program	

2081	**CRC/TNRC not allowed**	
Scan:	On processing in AUTOMATIC mode	
Effect:	• Processing interruption • NC START disabled	
Explanation:	• The alarm is displayed with the channel number and the block number • The following functions cannot be programmed with cutter radius/tool nose radius compensation selected: G33, G34, G35, G37, G58, G59, G92, G200, @714	
Remedy:	• Pre-program G40 • Cancel with G41/G42 D00 (CRC/TNRC)	

© Siemens AG 1991 All Rights Reserved

2082	**CRC plane not determinable**	
Scan:	On processing a part program in AUTOMATIC mode	
Effect:	• Processing interruption • NC START disabled	
Explanation:	• The alarm is displayed with the channel number and the block number • The axes for the selected CRC plane do not exist • G41/G42 has been selected and the plane definition (NC MD 548*, 550*, 552*) and/or G16 programming is incorrect.	
Remedy:	• MD 548* • MD 550* • Check MD 552* • Select correct plane with G16	

2087	**Coordinate rotation not allowed**	
Scan:	On processing in AUTOMATIC or MDA mode	
Effect:	• Processing interruption • NC START disabled	
Explanation:	• The alarm is displayed with the channel number and the block number • On selection of coordinate rotation (G54 to G59) a circular movement (G02, G03) has been programmed in the next block • The settable zero offset (G54 to G57) has been changed after selection of coordinate rotation (angle 0 degrees) • The plane (G16, G17, G18, G19) has been changed after selection of coordinate rotation (angle 0 degrees)	
Remedy:	• Correct program • The plane can only be changed if the angles of rotation and zero offsets programmed under G58 and G59 are zero • The settable zero offset and coordinate rotation (G54 to G57) can only be changed if the angles of rotation and zero offsets programmed under G58 and G59 are zero (Deselect if necessary)	

2089	**SERVO 1 computing time**	
Scan:	Cyclic	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready	
Explanation:	• The alarm is displayed with the serial number • The capacity of one servo CPU is not sufficient for the ratio of number of axes and sampling time set	
Remedy:	Check NC MD 155 and NC MDs 160 to 163 and use additional servo CPUs if necessary	

© Siemens AG 1991 All Rights Reserved

2105	SERVO 2 computing time	
Scan:	Cyclic	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready	
Explanation:	• The alarm is displayed with the serial number • The capacity of one servo CPU is not sufficient for the ratio of number of axes and sampling time set	
Remedy:	Check NC MD 155 and NC MDs 160 to 163 and use additional servo CPUs if necessary	

2121	SERVO 3 computing time	
Scan:	Cyclic	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready	
Explanation:	• The alarm is displayed with the serial number • The capacity of one servo CPU is not sufficient for the ratio of number of axes and sampling time set	
Remedy:	Check NC MD 155 and NC MDs 160 to 163 and use additional servo CPUs if necessary	

2137	SERVO 4 computing time	
Scan:	Cyclic	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready	
Explanation:	• The alarm is displayed with the serial number • The capacity of one servo CPU is not sufficient for the ratio of number of axes and sampling time set	
Remedy:	Check NC MD 155 and NC MDs 160 to 163 and use additional servo CPUs if necessary	

© Siemens AG 1991 All Rights Reserved

2160	Scale factor not allowed	
Scan:	On processing in AUTOMATIC or MDA mode	
Effect:	• Processing interruption • NC START disabled	
Explanation:	• The alarm is displayed with the channel number and the block number • The scale factor has exceeded the valid value range: – P is negative – P = 0 – P > 99.99999	
Remedy:	Observe valid value range for scale factor: (P = 0.00001 to 99.99999)	

2161	Scale change not allowed	
Scan:	On processing in AUTOMATIC or MDA mode	
Effect:	• Processing interruption • NC START disabled	
Explanation:	• The alarm is displayed with the channel number and the block number • With a scale factor > 1 an axis position has been programmed which is so great that internal representation is no more possible	
Remedy:	• Check programmed axis position • Reduce scale factor	

2165	Computing time exceeded	
Scan:	Cyclic	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay	
Explanation:	• The alarm is displayed with the serial number • The number of axes and spindles on the servo CPU is greater than the permissible maximum number. • The alarm is also set if the control recognizes the timing error, i. e. processing is no more performed in the IPO clock cycle. The set speeds are set to zero in the event of overrun of position control loops. All mode groups with axes on the relevant servo CPU are instantaneously stopped. The ”Computing time exceded ” alarm indicates that the servo CPU is overloaded.	
Remedy:	Report to Service	

© Siemens AG 1991 All Rights Reserved

2171	**Approach not possible**	
Scan:	On processing in AUTOMATIC or MDA mode	
Effect:	Processing interruption	
Explanation:	• The alarm is displayed with the channel number and the block number • No axis of the selected plane (G16, G17, G18, G19) has been programmed in the block following approach so that no vector can be calculated for tangential approach • @714 has been programmed in the selection block or in the subsequent block for soft approach	
Remedy:	• Correct program (G147, G247, G347) by programming at least **one** axis of the selected plane in the block following approach	

2172	**Retraction not possible**	
Scan:	On processing in AUTOMATIC or MDA mode	
Effect:	Processing interruption	
Explanation:	• The alarm is displayed with the channel number and the block number • No axis of the selected plane (G16, G17, G18, G19) has been programmed in the block preceding retraction so that no vector for tangential retraction can be calculated • @714 has been programmed in the deselection block or in the block before • G48 programmed **without** preceding selection	
Remedy:	• Correct program (G148, G248, G348, G48) by programming at least **one** axis of the selected plane in the block preceding retraction	

2173	**Wrong approach/retraction plane**	
Scan:	On processing in AUTOMATIC or MDA mode	
Effect:	Processing interruption	
Explanation:	• The alarm is displayed with the channel number and the block number • A change of plane is programmed in the block following selection (G16, G17, G18, G19) • A change of plane is programmed in the deselection block	
Remedy:	• Correct program (change of plane)	

© Siemens AG 1991 All Rights Reserved

2184	Illegal M function for C axis	
Scan:	• During ramp-up • On warm restart • On MD changes	
Effect:	The C axis drive for spindles on the 32 bit servo CPUs cannot be selected.	
Explanation:	• The alarm is displayed with the serial number • The numbers of the M functions for selecting and deselecting C axis drive are determined via MD 260 and MD 261. These numbers must not coincide with the defaulted M functions	
Remedy:	Valid values must be entered in MD 260 and MD 261	

2189	Transformation undefined	
Scan:	On selection of transformation	
Effect:	• NC START disabled • Processing interruption	
Explanation:	• Alarm is displayed with the block number • Type pf transformation not defined • Transformation axes are different mode groups • Option for selected transformation not available • Transformation selected in illegal channel • Transformation defined several times or wrongly defined • Transformation data block declared invalid by Alarm 3087 (error in transformation data)	
Remedy:	• Check transformation data block • Check program • Check channel number • Have option checked by Service and retrofit if necessary	

2190	Transformation axes assigned	
Scan:	On selection of transformation	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready	
Explanation:	• Alarm is displayed with the block number • A transformation has been selected whose real axes are also used in parallel-running transformation in another channel	
Remedy:	• Wait for deselection of transformation in parallel-running channel • Check program	

© Siemens AG 1991 All Rights Reserved

2191	**Transformation at zero**	
Scan:	On selection of transformation in AUTOMATIC (Block search) or MDA mode	
Effect:	• NC START disabled • Processing interruption • Block search is interrupted	
Explanation:	• Alarm is displayed with the block number • Transformation has been selected at a moment when one or several of the axes involved in transformation were in the actual position ZERO. • On selection of TRANSMIT the X axis (transverse axis) must not have the actual position ZERO. This also applies to block search.	
Remedy:	Set the real axes of the transformation to be selected to permissible actual positions prior to selecting transformation (on TRANSMIT the X axis to X 0)	

2192	**Axis is not a rotary axis**	
Scan:	In each G195 block	
Effect:	• NC START disabled • Cancellation of Mode Group Ready	
Explanation:	• Alarm is displayed with the block number • The axis name following G195 does not define a rotary axis	
Remedy:	Program axis name of rotary axis together with G195	

2194[d)]	**FIFO not available**	
Scan:	• On POWER ON • On warm restart	
Effect:	• Processing interruption • NC START disabled	
Explanation:	• The alarm is displayed with the serial number • FIFO channel has been assigned even though the FIFO option has not been activated.	
Remedy:	• Set option • Call service to check the function expansion and install is necessary.	

d) from GA2, Software Version 1

© Siemens AG 1991 All Rights Reserved

2195[d]	**Too many FIFO chan. defined**	
Scan:	• On POWER ON • On warm restart	
Effect:	• Processing interruption • NC START disabled	
Explanation:	• The alarm is displayed with the serial number. • A maximum of two channels per NC CPU can be assigned to the FIFO memory.	
Remedy:	Check MD 558*	

225* [c]	**Spindle speed too high**	
Scan:	Only with NC MD 520* set (encoder available)	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready	
Explanation:	• The alarm is displayed with the serial number. • The actual spindle speed is higher than that set in machine data or setting data	
Remedy:	• Program a smaller S value • NC MDs 403* to 410* (max. spindle speed for 1st to 8th gear stage) • NC MD 445* (tolerance band of max. spindle speed) • NC MD 451* (max. spindle speed) • Check PLC gear stage • Check G92 S... at "v=constant" • Check spindle speed limitation setting data • Program G26 S ...	

c) up to and including Software Version 6
d) from GA2, Software Version 1

© Siemens AG 1991 All Rights Reserved

226*c)	**Control loop spindle - Hardware** 
Scan:	Cyclic
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Servo enable of spindle is removed after the time in MD 477* has elapsed (waiting time for servo enable)
Explanation:	• The alarm is displayed with the serial number • The measuring circle differential signals – are not in phase – have a short-circuit to earth – are entirely missing
Remedy:	• Check whether measuring circuit connector has been plugged • You can check by inserting the measuring circuit short-circuiting plug whether the measuring-circuit module is in operating order • Check the differential signals using an oscilloscope • Replace sensor

227*c)	**Spindle measuring system dirty** 
Scan:	Cyclic
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready
Explanation:	• The alarm is displayed with the serial number • On measuring systems with a contamination signal, the measuring system has signalled a fault to the NC
Remedy:	Check the measuring system

228*c)	**M19 option not available** 
Scan:	On processing in AUTOMATIC or MDA mode or when processing an external program is specified
Effect:	Function is not executed
Explanation:	• The alarm is displayed with the serial number • M19 has been programmed, although this function is not implemented in the control
Remedy:	• Check part program or PLC user program • Check NC MD • Have option checked by Service and retrofit if necessary

c) up to and including Software Version 6

© Siemens AG 1991 All Rights Reserved

229*c)	**Parameter assignment error**	
Scan:	• During ramp-up • On warm restart • On MD changes	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • A service number is shown in the diagnostics display of the spindle concerned	
Explanation:	Errors are ascertained on checking the spindle machine data or the associated C axis	
Remedy:	Check the following machine data and correct if necessary: • MD 155 • MDs 160 to 163 • MD 168 • MD 200*, MD 384* • MD 364*, MD 368* • MD 400*, MD 460* • MDs 403* to 410* • MDs 419* to 426* • MDs 478* to 485* • MD 435* • MD 455*, MD 456* • MD 458* • MD 459* • MD 461* • MD 463* • MD 466* • MD 468* • MD 1800* A service number is shown in the diagnostics display of the spindle concerned. You can see the meaning of the service number from the ”Extended Spindle Functions” Function Manual.	

c) up to and including Software Version 6

© Siemens AG 1991 All Rights Reserved

230*c)	**Function cannot be executed** 
Scan:	• On programming the M function for selecting or deselecting C axis operation • On programming the G function for selecting or deselecting synchronous operation
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready
Explanation:	C axis operation or synchronous operation has been selected or deselected for a spindle which is defined on a 16 bit servo CPU
Remedy:	• Check assignment of spindle and axis to the servo CPU (MD 400*, 200*, 461*) • Check the part program

231*c)	**Illegal NC command** 
Scan:	On selecting M19
Effect:	• Processing interruption • NC START disabled
Explanation:	M19 has been requested for a spindle which is in C axis or synchronous operation
Remedy:	• Check the part program • Check the PLC program (M19 of PLC)

c) up to and including Software Version 6

© Siemens AG 1991 All Rights Reserved

2410	**FA1 in controlled coupling**	
2411	**FA2 in controlled coupling**	
2412	**FA3 in controlled coupling**	
Scan:	Cyclic, when the electronic gear (ELG) is active or when the ”Synchro-spindle” function has been selected	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Only the actual values of the LA/LS are used for determining the set-points of the FA/FS. • The coupling between LA/LS and FA/FS is maintained at first	
Explanation:	• The alarm is displayed with the serial number • An error of the LA/LS or other axes or spindles of the same mode group has occurred. The coupling is maintained as long as the FA/FS itself is not faulted. This is only guaranteed for a certain time which is specified in MD 956/976/996.	
Remedy:	Clear fault with the LA/LS or the concerned axis or spindle within the mode group. The coupled axes/spindles should be separated.	

© Siemens AG 1991 All Rights Reserved

2413	**FA1 Emergency retraction**	
2414	**FA2 Emergency retraction**	
2415	**FA3 Emergency retraction**	
Scan:	Cyclic, when the electronic gear (ELG) is active or when the ”Synchro-spindle” function has been selected and emergency retraction monitoring has been enabled	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • The emergency signal EMERGENCY RETRACTION ACTIVE is set • When using the mixed I/O module, a respective hardware signal is set	
Explanation:	• The alarm is displayed with the serial number • The synchronism between LA/LS and FA/FS is continuously monitored. If the deviation exceeds the emergency retraction threshold (MD 954/974/994), the alarm is activated if emergency retraction monitoring has been enabled (emergency signal or SD bit).	
Remedy:	• Is emergency retraction enable (emergency signal of SD bit) desired? • Check drive of FA/FS and LA/LS • Check limiting values for speed and acceleration • Check emergency retraction threshold • Check the transmission ratios	

© Siemens AG 1991 All Rights Reserved

2416	**FA1 Speed limitation**	
2417	**FA2 Speed limitation**	
2418	**FA3 Speed limitation**	
Scan:	Cyclic, when the electronic gear (ELG) is active or when the ”Synchro-spindle” function has been selected	
Effect:	• Processing interruption • NC START disabled • The speed of the FA/FS is limited to its maximum value • The setpoint determined by the balancing controller is ignored during limitation	
Explanation:	• The alarm is displayed with the serial number • The setpoint calculated for the FA/FS has exceeded the maximum speed of the FA/FS. Synchronism between LA/LS and FA/FS is at risk.	
Remedy:	• Reduce the speed of the LA/LS • Check limiting values for the speed of the FA/FS • Check the transmission ratios	

2419	**FA1 Acceleration limitation**	
2420	**FA2 Acceleration limitation**	
2421	**FA3 Acceleration limitation**	
Scan:	Cyclic, when the electronic gear (ELG) is active or when the ”Synchro-spindle” function has been selected and the MD bit ”Suppression of acceleration limitation ” is not set	
Effect:	• Processing interruption • NC START disabled • If the ”Acceleration limitation synchronous” signal (emergency signal, SD bit) is set, traversing is continued at maximum acceleration. Set part positions suppressed during the movement are subsequently traversed to. • The setpoint speed determined by the balancing controller is ignored during limitation to maximum acceleration	
Explanation:	• The alarm is displayed with the serial number • The acceleration calculated for the FA/FS has exceeded the maximum acceleration of the FA/FS. Synchronism between LA/LS and FA/FS is at risk.	
Remedy:	• Reduce the speed or the acceleration of the LA/LS • Check the limiting value for acceleration of the FA/FS • Check the transmission ratio • Suppress acceleration limitation if necessary (MD bit)	

© Siemens AG 1991 All Rights Reserved

2422	**FA1 Configuration error**	
2423	**FA2 Configuration error**	
2424	**FA3 Configuration error**	
Scan:	• During ramp-up • On warm restart • On MD changes	
Effect:	• NC START disabled • The incorrectly configured LA/LS is ignored	
Explanation:	• The alarm is displayed with the serial number • The following checks are made: – Are the real LA/LS assigned to a measuring circuit? – Is the FA/FS assigned to a measuring circuit? – Is one axis/spindle defined as FA/FS more than once? – Has the FA/FS itself as LA/LS? – Is there any concatenation of LA/LS and FA/FS?	
Remedy:	• Check the measuring circuit assignment • Configure LA as a fictitious LA or assign the LA to a measuring circuit • Check the configuration of LA/LS and FA/FS	

2425	**FA1 Suppression inhibited**	
2426	**FA2 Suppression inhibited**	
2427	**FA3 Suppression inhibited**	
Scan:	Cyclic, when the electronic gear (ELG) is active or when the ”Synchro-spindle” function has been selected and a suppressing travel command for the FA/FS is given	
Effect:	• Processing interruption • NC START disabled • The suppressing traversing movement of the FA/FS is not performed • The coupling is maintained	
Explanation:	• The alarm is displayed with the serial number • A suppressing traversing movement of the FA/FS is only possible if suppression has been enabled (Emergency signal or SD bit). Suppression must also be enabled for the functions: – ”In-process synchronisation” (ELG) – ”Semi-automatic centering” (ELG) – ”Defined angle offset” (synchro-spindle)	
Remedy:	• Set emergency signal or SD bit ”FA/FS suppression On” or • Avoid suppression	

© Siemens AG 1991 All Rights Reserved

3000	**General programming error**	
Scan:	On processing a part program or program block in AUTOMATIC or MDA mode	
Effect:	Processing interruption	
Explanation:	• The alarm is displayed with the channel number and the block number • In one block of the program a general, not precisely definable, programming error has been made • Division by 0 • A non-existent G function has been programmed • Non-existent R parameters have been programmed • No +, -, /, * has been programmed with R parameter chaining • Value range exceeded with T parameter calculation • Number of decades exceeded (M, S, T, D, H, L, P, F) • Number of subroutine passwords P not programmed directly behind L • Main block":" programmed in the subroutine • 2 decimal points programmed • Decimal point programmed with M, S, T, D, H, L, P • More than 8 decades programmed	
Remedy:	• Check the faulty block in "Correction block" display • If possible, the cursor is positioned in front of the word containing the error • The number of the block containing the error is displayed in the alarm line after the alarm no.	

3001	**Geometry parameters > 5**	
Scan:	On processing part programs	
Effect:	• Processing interruption	
Explanation:	• The alarm is displayed with the channel number and the block number • More than 2 radii or 2 angles have been programmed in the block • More than 5 geometry parameters, such as axes, interpolation parameters, radii, angles ... have been programmed in the block	
Remedy:	• Check the faulty block in "Correction block" display • If possible, the cursor is positioned in front of the word containing the error • The number of the block containing the error is displayed in the alarm line after the alarm no.	

© Siemens AG 1991 All Rights Reserved

3002	**Polar/radius error**
Scan:	On processing part programs
Effect:	Processing interruption
Explanation:	• The alarm is displayed with the channel number and the block number • In the block with polar/radius programming, no programming for: – Angle – Radius • Full circle – Coordinates for the centre point not specified as absolute values
Remedy:	• Check the faulty block in ”Correction block” display • If possible, the cursor is positioned in front of the word containing the error • The number of the block containing the error is displayed in the alarm line after the alarm no.

3003	**Invalid address**
Scan:	On processing in AUTOMATIC or MDA mode
Effect:	Processing interruption
Explanation:	• The alarm is displayed with the channel number and the block number • A different address has been programmed from that defined in NC MD or the plane assignment (cube) and the axis definition in the channel do not agree Example: Incorrect: **N20 G0 C100 L_F** (rapid traverse 4th axis) In NC MD the 4th axis has been defined with adress Q Correct: **N20 G0 Q100 L_F** • The programmed axes are not defined in the selected mode group (with G200)
Remedy:	• Check the faulty block in ”Correction block” display • Check the channel-specific MD 548* • If possible, the cursor is positioned in front of the word containing the error • The number of the block containing the error is displayed in the alarm line after the alarm no.

© Siemens AG 1991 All Rights Reserved

3004	CL800 error	
Scan:	Cyclic	
Effect:	Processing interruption	
Explanation:	The alarm is displayed with the channel number and the block number Formal errors • wrong characters entered (0...9, a...f are permitted) • @-number greater than 3 decades • @-number or @-function not implemented in 880 • @-number or @-function not programmable or defined in CL800 Input error with address letters and numerical values • Incorrect address letters (K, R, P are permitted) • Number of decades too high (max. K 8 decades permitted) • Number of decades too high (max. R 4 decades permitted) • Number of decades too high (max. P 4 decades permitted) • R parameter number not defined or too high • Point programmed in R parameter number • Point programmed in P parameter number • Incorrect number of words Input error with specific @-functions: Program branching • Error in the block number (programmed point, block number greater than 4 decades) Data transfer system cell - R parameter • Constant or R parameter contents programmed too large for information such as: axis number, channel number, TO area, NC/PLC machine data, NC setting data, D number, P number, group for zero offsets, default "COARSE/FINE" alarm number • Bit number too large (0 to 7 are permitted) • System cell non-existent • Incorrect value input for system cell Mathematical and logical functions • Value selected too high for square root (+ 00 000 001...99 999 999 permitted) • Incorrect angle selected for sine (-360 (0) +360 permitted) • Two constants used for: Angle from two vector components, OR, EXOR, AND, NAND • Incorrect sign input for logical functions (0, 1 permitted) (only bits and bytes) (max. 8 bits) NC-specific functions • Incorrect address letter used for number of axes • Number of axes selected too high (max. three axes permitted per block) • No axis name programmed (0)	

© Siemens AG 1991 All Rights Reserved

3005	**Contour definition error**	
Scan:	On processing in AUTOMATIC mode	
Effect:	• Processing interruption	
Explanation:	• The alarm is displayed with the channel number and the block number • The coordinates in blueprint programming have been defined so that no intersection point is given • Too many geometry values have been programmed	
Remedy:	• Check the faulty block in ”Correction block” display • If possible, the cursor is positioned in front of the word containing the error • The number of the block containing the error is displayed in the alarm line after the alarm no.	

3006	**Wrong block structure**	
Scan:	On processing part programs in AUTOMATIC or MDA mode	
Effect:	Processing interruption	
Explanation:	• The alarm is displayed with the channel number and the block number • Approach to reference point in the program and specification of wrong G function or more than 1 axis or an illegal axis • Incorrect thread lead parameters with G33/G37 • More than 2 axes are programmed with G33/G34/G35 • More than 3 M functions in the block • More than 1 S function in the block • More than 1 T function in the block • More than 1 H function in the block • More than 4 auxiliary functions in the block • More than 6 axes+geometry parameters • More than 5 axes with G00/G01/G02/G03/G200 • More than 2 axes with G10/G11 • More than 2 axes with G02/G03 (circular parameter programming) • G04 programmed with addresses other than X, F or S • Not only G04 programmed in the block • M19 S programmed with other functions • Incorrect circular parameters with G02/G03 axes • Axis missing with circle radius programming • G10/G11 block must be programmed in front of the 1st G110 block in the program • G110 must not be programmed with axes • Incorrect programming in conjunction with G176 freezing function • G200 programmed with other functions • A traversing movement has been programmed with G91 immediately after a block with G74 (Reference point approach in program) or with G130/G230/G330 (Transformation) or with @720 (Measuring). Only G90 is permissible. • If cylindrical interpolation is active, the circular contours must not be programmed with interpolation parameters.	
Remedy:	• Check the faulty block in ”Correction block” display • If possible, the cursor is positioned in front of the word containing the error • The number of the block containing the error is displayed in the alarm line after the alarm no.	

© Siemens AG 1991 All Rights Reserved

3007	Wrong setting data program	
Scan:	On processing program blocks in AUTOMATIC or MDA mode	
Effect:	Processing interruption	
Explanation:	• The alarm is displayed with the channel number and the block number • M19 without S word • Spindle not available • Illegal setting data, e.g. G25, G26, G92 X. . . Y. . ., G92 D. . ., T. . ., I. . . or J. . .	
Remedy:	• Check the faulty block in ”Correction block” display • If possible, the cursor is positioned in front of the word containing the error • The number of the block containing the error is displayed in the alarm line after the alarm no.	

3008	Subroutine error	
Scan:	On processing programs in AUTOMATIC or MDA mode	
Effect:	Processing interruption	
Explanation:	• The alarm is displayed with the channel number and the block number • M17 not in subroutine • M02/M30 in subroutine • M17 in subroutine • More than 3 subroutine levels • MD 284 = 0 with G203/G204	
Remedy:	• Check the faulty block in ”Correction block” display • If possible, the cursor is positioned in front of the word containing the error • The number of the block containing the error is displayed in the alarm line after the alarm no.	

3009	Program disabled	
Scan:	• On NC START • When editing a program during processing	
Effect:	Function is not executed	
Explanation:	• The alarm is displayed with the channel number and the block number • A program is called with NC START which has been disabled by ”Opening”, "Copy" or "Rename". A program being edited may not be called with NC START. • If a program is to be edited while another program is being processed it must be disabled	
Remedy:	On completion of editing the disabled program must be enabled	

© Siemens AG 1991 All Rights Reserved

3010	**Intersection error**	
Scan:	On processing part programs in AUTOMATIC mode	
Effect:	Processing interruption	
Explanation:	• The alarm is displayed with the channel number and the block number • An error has been ascertained during reference preparation in conjunction with calculation of the intersection point caused by: – Contour program without G00, G01, G03 – Contour program with "Clear buffer memory" (@ 714) – Programmed axes different from selected plane – No intersection found – Circular cutting path – R parameter number non-existent – Incorrect plane assignment	
Remedy:	Check program containing the contour	

3011	**Axis twice/axis number > 5**	
Scan:	On processing part program blocks in AUTOMATIC or MDA mode	
Effect:	Processing interruption	
Explanation:	• The alarm is displayed with the channel number and the block number • An axis has been programmed twice in the same block • More than five axes have been programmed	
Remedy:	• Check the faulty block in ”Correction block” display • If possible, the cursor is positioned in front of the word containing the error • The number of the block containing the error is displayed in the alarm line after the alarm no.	

3012	**Block not in memory**	
Scan:	On block search or jumps in the part program	
Effect:	Processing interruption	
Explanation:	• The alarm is displayed with the channel number and the block number • The block number has not been found in the program in a block search • In a jump in the program the programmed block number could not been found • The program has not been terminated with M30, M17	
Remedy:	• Check the part program for correct block number destination or correct program termination with M30/M02 or M17	

© Siemens AG 1991 All Rights Reserved

3013	**Simulation data not enabled**	
Scan:	While simulation is active	
Effect:	Simulation is stopped	
Explanation:	• The alarm is displayed with the channel number and the block number • Data have been accessed in the simulated part program that are disabled for simulation	
Remedy:	• Check program • Use R0 to R199 with @ commands instead of R900 to R999 • Check NC MD 5009 bit 4	

3014	**Axis blocked (disabled) in channel**	
Scan:	On processing program blocks in AUTOMATIC or MDA mode or when processing external programs is specified	
Effect:	Processing interruption	
Explanation:	• The alarm is displayed with the channel number and the block number • The programmed axis has been disabled for this channel via NC MD 576*	
Remedy:	• Pay attention to the machine tool manufacturer's Programming Instructions • Correct machine data if necessary	

3015	**Main block not in memory**	
Scan:	On automatic block search	
Effect:	Machining is not started	
Explanation:	• The alarm is displayed with the alarm number and the block number • No main block is found in front of the target block during automatic block search	
Remedy:	• Check block found • Use different block search	

3016	**External data input error**	
Scan:	On data input from PLC to NC	
Effect:	Processing interruption	
Explanation:	• The alarm is displayed with the serial number • For external data input from PLC and NC – The code is incorrect – The value is too large – The dimension identifier is invalid	
Remedy:	• Check PLC program	

© Siemens AG 1991 All Rights Reserved

3017	Part program number occurs twice	
Scan:	On POWER ON	
Effect:	Processing interruption	
Explanation:	• The alarm is displayed with the serial number • There is a part program present on the UMS submodule which is also present in the part program memory of the NC (with the same identifer)	
Remedy:	• Delete the program in the part program memory or rename • Use a different UMS	

3018	Distance to contour too large	
Scan:	On AUTO interruption in contour block and departure from interruption point	
Effect:	• Processing interruption • Supplementary alarm 2048	
Explanation:	• The alarm is displayed with the channel number and the block number • This alarm is triggered when automatic operation is interrupted while a contour block is being executed and when the axes are positioned in an area, e. g. for tool change, that is outside the permissible tolerance for repositioning (scratch) (MD9). In order to avoid wrong positioning, the supplementary alarm 2048 (circle point error) is set. • Repositioning is only possible after Reset	
Remedy:	• Cancel alarms and activate block search to the interruption point • Check whether MD9 value can possibly be increased	

3019	Option RS232C (V.24) not available	
Scan:	On selection of interface	
Effect:	Function is not executed	
Explanation:	• Alarm is displayed with the block number • The 2nd RS232C (V.24) interface has been started either from the PLC or with the softkey, but the option is not fitted	
Remedy:	• Transfer data via the 1st RS232C (V.24) interface • Have option checked by Service and retrofit if necessary	

© Siemens AG 1991 All Rights Reserved

3020	**Option not available**	
Scan:	On specifying a function that has not been implemented	
Effect:	The function is not executed	
Explanation:	• Alarm is displayed with the block number • A function has been programmed or selected that is not implemented in the control	
Remedy:	Have option checked by Service and retrofit if necessary	

3021	**CRC/TNRC contour error**	
Scan:	With CRC/TNRC selected NOT: – in selection block – in deselection block	
Effect:	MD 5024.0=0: Machining is not interrupted, alarm can be acknowledged MD 5024.0=1: Machining is not interrupted. The operator can either stop machining (NC RESET) or acknowledge the alarm and then continue machining with NC START	
Explanation:	• Alarm is displayed with the channel number and the block number • The compensation calculation results in a traversing movement which is opposite to the one programmed • No intersection is obtained with linear-circular or circular-circular transitions in the angle range > 180°	
Remedy:	Check programming and tool radius	

3022	**More than one spindle programmed**	
Scan:	On processing part programs in AUTOMATIC or MDA mode	
Effect:	The function is not executed	
Explanation:	• The alarm is displayed with the serial number • Only one spindle can be programmed in one part program block	
Remedy:	Split spindle programming into two or more blocks	

© Siemens AG 1991 All Rights Reserved

3024	Display description not available	
Scan:	On display selection	
Effect:	The display selected does not appear	
Explanation:	• The alarm is displayed with the serial number • An attempt has been made with a programmed softkey to select a display that does not exist in the UMS or system memory	
Remedy:	Check the configured display number and the configured softkey function using the programming workstation	

3025	Display description error	
Scan:	On display selection	
Effect:	The display selected does not appear	
Explanation:	• The alarm is displayed with the serial number • A display has been programmed with graphic elements although the control has no "Graphics" option • The display selected has too many variables or fields • A type of display has been programmed which the control does not recognize	
Remedy:	• Check the display using the programming workstation • Have option checked by Service and retrofit if necessary	

3026	Graphics/text too voluminous	
Scan:	On display selection via softkey	
Effect:	The display contents is not displayed completely	
Explanation:	• The alarm is displayed with the serial number • Configuring error in the display selected • Sum of graphics and text elements too large	
Remedy:	• Check the display using the programming workstation • Split the contents over two displays if necessary	

3027	Graphics command too voluminous	
Scan:	On display selection via softkey	
Effect:	No graphics display	
Explanation:	• The alarm is displayed with the serial number • Sum of the graphics commands in display selected too large	
Remedy:	• Check the display using the programming workstation • Split the contents over two displays if necessary	
Caution:	This alarm causes alarm 3026 to be triggered	

© Siemens AG 1991 All Rights Reserved

3028	**Too many fields/variables**	
Scan:	On display selection via softkey	
Effect:	The display selected does not appear	
Explanation:	• The alarm is displayed with the serial number • Programming error in the display selected. The number of fields or variables is limited. A maximum number of fields/variables cannot be specified as the fields/variables may have differing formats and lengths	
Remedy:	• Check the display using the programming workstation • Reduce the number of fields/variables • Split the contents over two displays if necessary	

3029	**Graphics option not available**	
Scan:	On display selection via softkey	
Effect:	No graphics display	
Explanation:	• The alarm is displayed with the serial number • A display has been programmed with graphic elements although the control has no "Graphics" option	
Remedy:	• Program displays without graphic elements • Have option checked by Service and retrofit if necessary	

3030	**Cursor memory not available**	
Scan:	On display selection via softkey	
Effect:	The display selected is treated as if no cursor were available	
Explanation:	• The alarm is displayed with the serial number • The cursor memory programmed for the display selected is not correct (number not allowed or too large)	
Remedy:	Redefine the cursor memory using the programming workstation	
Caution:	The function of the cursor memory is to position the cursor where it was previously when the display is called again	

© Siemens AG 1991 All Rights Reserved

3032	Too many fields/variables	
Scan:	Cyclic	
Effect:	Not all variable values/texts are displayed	
Explanation:	• The alarm is displayed with the serial number • The programmed variable display section of the display description is too voluminous	
Remedy:	• Check the display using the programming workstation, reconfigure if necessary • Reduce variable display part	

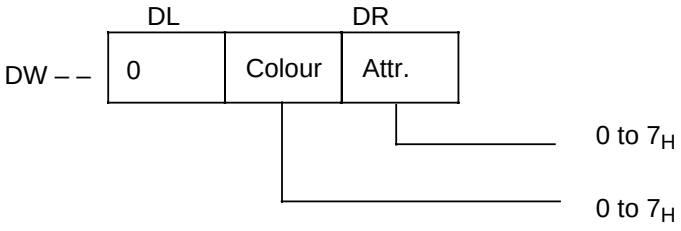
3033	Display text not available	
Scan:	Cyclic	
Effect:	The display text does not appear	
Explanation:	• The alarm is displayed with the serial number • The display text prepared at the programming workstation has not been transferred to the link list	
Remedy:	Check link list and re-link using the programming workstation. Watch out for linking errors.	

3034	Text not available	
Scan:	On display selection via softkey	
Effect:	The special text is not displayed	
Explanation:	• The alarm is displayed with the serial number • In the display selected the following texts were either linked incorrectly or not linked: – Menu texts – Interactive dialog texts – Mode texts – Alarm texts	
Remedy:	Check the display using the programming workstation and reconfigure if necessary	

© Siemens AG 1991 All Rights Reserved

3035	**Ind. addressing incorrect**	
Scan:	Cyclic	
Effect:	Variable values/texts are not displayed	
Explanation:	• The alarm is displayed with the serial number • The display description for indirect addressing is not correct. This refers to the specifications in the display heading for: – Data group – Data type – Data number – Data block • Specified number of indirect elements of list/display (EL) is not correct • Variable text (VT) selected, but status not deselected or offset not deselected • Variable value (VV) selected, but status not deselected	
Remedy:	• Check specifications for – Range start pointer – Range length pointer – Cursor and check their interrelationship • Check deselection of status or offsets	

3036	**Variable status incorrect**	
Scan:	Cyclic	
Effect:	The status indicated in the display description is displayed	
Explanation:	• The alarm is displayed with the serial number • An illegal status is read Illegal: Low Nibble (bits 0 to 3) > 7_H High Nibble (bits 4 to 7) > 7_H • An illegal display status is specified by the PLC	
Remedy:	The status specified by the PLC user program must be corrected	



© Siemens AG 1991 All Rights Reserved

3040	Fields/variable not displayable	
Scan:	Cyclic	
Effect:	Not all variable values/texts are displayed	
Explanation:	• The alarm is displayed with the serial number • Field/variable incorrectly configured (data group not available) • Field/variable configured with too few characters • Field/variable overflow (value range exceeded) • Format error, format cannot be changed • Illegal pointer • If the alarm occurs with standard displays, the value range has been exceeded	
Remedy:	Check field/variable using the programming workstation, delete and re-enter if necessary	

3041	Too many fields/variables	
Scan:	Cyclic	
Effect:	Not all variable values/texts are displayed	
Explanation:	• The alarm is displayed with the serial number • The display description has been made too voluminous, the internal buffer memory is not large enough to process the display • Too many variable values or texts have been programmed • A maximum number of fields/variables cannot be specified as the fields/variables have differing formats and lengths	
Remedy:	The desired information must be reduced or split over several displays using the programming workstation	

3042	Display description error	
Scan:	On display selection via softkey	
Effect:	The display selected does not appear	
Explanation:	• The alarm is displayed with the serial number • An error has been found in the display description which cannot be exactly defined, e. g. a non-existent field has been programmed (NC MD for 33rd axis with 880)	
Remedy:	Check the display using the programming workstation	

© Siemens AG 1991 All Rights Reserved

3044	**Graphics subroutine too large/missing**	
Scan:	Cyclic	
Effect:	No variable values/texts are displayed	
Explanation:	• The alarm is displayed with the serial number • An error has occurred in the system program or graphics processor as regards the variable display design	
Remedy:	Replace system program	

3045	**Graphics subroutines used too large**	
Scan:	On display selection via softkey	
Effect:	No graphics display	
Explanation:	• The alarm is displayed with the serial number • The total of the graphics subroutines used in the display is too high	
Remedy:	• Simplify display description • Use arcs instead of polygon curves	

3046	**Variable fault**	
Scan:	On display selection via softkey	
Effect:	Not all variable values/texts are displayed	
Explanation:	• The alarm is displayed with the serial number • A variable text **without** end identifier has been programmed in display description caused by an error in the programming workstation software. A transfer format error is the result.	
Remedy:	Check programming workstation (error must be located at the interruption point of the following elements that are no longer displayed)	

3050	**Wrong input**	
Scan:	During the simulation function, when reading tool offsets	
Effect:	The function is not executed	
Explanation:	• The alarm is displayed with the serial number • Number of tool probes too small/large • Tool type > 39 or < 0 (possible values: 0 to 39) • Empirical value number too large • Measuring parameter > 2 or < 0 (possible values: 0 to 2)	
Remedy:	Check simulation data or tool offsets and correct values if necessary	

© Siemens AG 1991 All Rights Reserved

3051	Programmed feed missing/wrong	
Scan:	While simulation is active	
Effect:	Simulation is not stopped	
Explanation:	• The alarm is displayed with the channel number and the block number • For graphic simulation a program is simulated which does not contain a feed value in front of the 1st traversing block	
Remedy:	Correct program	

3052	Simulation error	
Scan:	While simulation is active	
Effect:	Simulation is not stopped	
Explanation:	• The alarm is displayed with the channel number and the block number • Internal simulation error	
Remedy:	Have error removed by Service	

3054	Program not simul. correctly	
Scan:	While simulation is active	
Effect:	None, simulation continues	
Explanation:	• The alarm is displayed with the channel number and the block number • A function has been programmed that cannot be properly run in simulation – Programmed plane G16 does not agree with a standard plane (G17/G18/G19) – Synchronisation has been lost with dual saddle simulation by means of G04 – G06/G68/G74 cannot be simulated – Coupled motion cannot be simulated – Transformation cannot be simulated	
Remedy:	None	

© Siemens AG 1991 All Rights Reserved

3058	**VTE 340 cannot be aborted**	
Scan:	Each time terminal emulation is deselected	
Effect:	Terminal emulation cannot be terminated in this mode (return to NC operation not possible)	
Explanation:	• The alarm is displayed with the serial number • Exit from emulation is not permitted while it is in setup mode , in graphics mode or in local edit mode	
Remedy:	• Exit from setup mode of the terminal emulator • Exit from graphics mode of the terminal emulator • Exit from local edit mode of the terminal emulator	

3059	**Setup cannot be called**	
Scan:	Each time setup mode is selected	
Effect:	Setup mode cannot be selected within terminal emulation	
Explanation:	• The alarm is displayed with the serial number • Setup mode must not be selected as long as the terminal emulation is in graphics mode	
Remedy:	Exit from the graphics mode of the terminal emulation	

3060[d)]	**VTE 340 cannot be called**	
Scan:	On terminal emulation	
Effect:	Terminal emulation is not possible.	
Explanation:	• The alarm is displayed with the serial number • The wrong interface was parameterized when terminal emulation was called.	
Remedy:	• Parameterize the correct interface	

d) from GA2, Software Version 1

© Siemens AG 1991 All Rights Reserved

3061[d)]	**Fault in keyboard**	
Scan:	On POWER ON and cyclic	
Effect:	The operator panel and the full standard keyboard (if connected) are not working	
Explanation:	This error message is triggered if, for example, the reset key on the keyboard interface module is activated or the connection between the operator panel interface module and the INTKEY module breaks down during operation.	
Remedy:	Check the connection between the operator panel interface and the INTKEY module, then execute a POWER ON.	

3072	**Message not available**	
Scan:	Cyclic	
Effect:	None	
Explanation:	• The alarm is displayed with the serial number • Alarms are provided in cycle preparation for which no text has been programmed • NC alarms occur for which no text is provided in the system • Study list of alarm displays and check for alarm numbers without text	
Remedy:	• Program text for cycle alarms • Send a message to the system management in the event of system alarms	

3077[d)]	**Wrong rotary axis**	
Scan:	On every block change	
Effect:	• Processing interruption • NC START disabled	
Explanation:	• The alarm is displayed with the channel number and the block number • A rotary axis has been programmed with G203/G204 which is not assigned to any spindle	
Remedy:	Assign the C axis number to a spindle in MD 461*	

d) from GA2, Software Version 1

© Siemens AG 1991 All Rights Reserved

3078[d)]	**Wrong infeed axis**	
Scan:	On every block change or in the tapping selection block	
Effect:	• Processing interruption • NC START disabled	
Explanation:	• The alarm is displayed with the channel number and the block number • The infeed axis is not in the G16/G19 plane	
Remedy:	The infeed axis must be perpendicular to the current plane	

3079[d)]	**No S value for thread**	
Scan:	On every block change or in selection block	
Effect:	• Processing interruption • NC START disabled	
Explanation:	• The alarm is displayed with the channel number and the block number • No S value has been programmed in the G203/G204 selection block.	
Remedy:	Program S value	

3081	**CRC/TNRC not selected for approach**	
Scan:	On processing in AUTOMATIC or MDA mode	
Effect:	• NC START disabled • The program is executed without approach element	
Explanation:	• The alarm is displayed with the channel number and the block number • CRC/TNRC (G41/G42) has not been selected in or in front of the approach block • CRC/TNRC has not been deselected in or after the approach block	
Remedy:	• Soft approach/retraction is only possible when CRC/TNRC has been selected since otherwise the programmed compensating movement cannot be precisely calculated • Correct program (G41/G42) • Program D0 with G41/G42 if necessary	

d) from GA2, Software Version 1

© Siemens AG 1991 All Rights Reserved

3084	**Illegal working area limitation**	
Scan:	Cyclic	
Effect:	The control automatically enters the maximum value permitted by the traversing range in the working area limitation	
Explanation:	• The alarm is displayed with the block number • A value has been entered in the minimum or maximum axis-specific working area limitation that lies outside the permissible traversing range of the relevant axis	
Remedy:	• Check input • Check program (G25/G26, @...) • Look up maximum traversing range in table (combination of axis-specific position control resolution and input resolution)	

3085	**NC CPU time watch-dog**	
Scan:	Cyclic	
Effect:	None, brief irregular axis motions may occur	
Explanation:	• The alarm is displayed with the block number • Time overload in the NC CPU as a result of parallel program operation or due to selected functions such as – Transformation (TRANSMIT, ...) – Coupled motion	
Remedy:	Stop program with NC STOP	

3086	**Illegal transformation selection**	
Scan:	On selection/deselection of transformation via the command channel from the PLC	
Effect:	NC START disabled	
Explanation:	• The alarm is displayed with the serial number • The PLC has transferred an illegal value via the command channel. This is evaluated in the error byte in the PLC.	
Remedy:	Check PLC user program	

© Siemens AG 1991 All Rights Reserved

3087	Error in transformation data	
Scan:	• On POWER ON • On warm restart	
Effect:	The transformation data block selected is disabled	
Explanation:	• The alarm is displayed with the block number • An illegal value has been assigned to one NC MD in the transformation data block selected. The illegal NC MD number is entered in the block number of alarm 3087	
Remedy:	Check NC MD for transformation data (block number of alarm 3087)	

3089	KV factor for thread too small	
Scan:	When thread is programmed	
Effect:	Calculation is terminated if the calculated K_V factor exceeds the limit under consideration of the IPO time	
Explanation:	• The alarm is displayed with the serial number • The input K_V factor is too small	
Remedy:	Enter larger servo gain factor, see also Installation Instructions, Function Description, "Extended Thread Package".	

3090	Following axis programmed	
Scan:	Cyclic	
Effect:	None	
Explanation:	• The alarm is displayed with the serial number • Alarm 3090 is output if an axis which has been defined as a folllowing axis is to be traversed in JOG mode on its own	
Remedy:	• Other definition of the coupled axis grouping to prevent this axis from being a following axis • Deselect coupled motion function	

© Siemens AG 1991 All Rights Reserved

3091	**Reduction at SW prelimit switch** 
Scan:	In AUTOMATIC or MDA mode when processing a part program block or when positioning the axes in JOG mode
Effect:	Reduce speed to the value set in the machine data
Explanation:	• The alarm is displayed with the block number • The software prelimit switch has been overrun, the axis speed is decelerated to the reduced speed (MD 1)
Remedy:	• Check the traversing block • Check the value in NC MD 1100 (prelimit switch) • Position axis outside the prelimit switch range and cancel the alarm by acknowledging it

3092	**Programmed speed too high** 
Scan:	On processing programs in AUTOMATIC or MDA mode
Effect:	None
Explanation:	• The alarm is displayed with the block number • The specified speed (either programmed or via override setting) is higher than the path speed resulting from the maximum axis speeds
Remedy:	• Program slower path speed or check override • Check NC MD 280* (maximum speed) • Program lower spindle speed with G95

3093[d)]	**G171 illegal** 
Scan:	On processing programs in AUTOMATIC or MDA mode
Effect:	• Processing interruption • NC START disabled
Explanation:	• The alarm is displayed with the channel number and the block number • No FIFO has been assigned to the current channel • The FIFO option has not been activated
Remedy:	• Correct program • Check MD

d) from GA2, Software Version 1

© Siemens AG 1991 All Rights Reserved

3094[d)]	**Error in compensation data**	
Scan:	• On POWER ON • On warm restart	
Effect:	IKA compensation data are not used in the calculations	
Explanation:	• The alarm is displayed with the block number • The conversion of the error curves for IKA resulted in an error • The following was also output: - Channel number contains error number block number (N....) contains error location - Error no. 1: Only one or other of the start and end pointers is 0. Both must be either 0 or < > 0. Block number = no. of error curve - Error no. 2: End pointer is not greater than start pointer Block number = no. of error curve - Error no. 3: Within one error curve section, the actual position (n + 1) is smaller than the actual position (n) Block number = number of compensation point - Error no. 4: Within one error curve section, the actual position (n + 1) is equal to the actual position (n) Block number = number of compensation point - Error no. 5: Straight gradient within an error curve section is 45 degrees Block number = no. of compensation point **Example:** Alarm line: "**`3094 3 N105`** error in compensation data": In the case of compensation point 105, the actual position 106 is less than the actual position 105.	
Remedy:	• Check and correct IKA data • Carry out POWER ON Start or warm restart	

3098[d)]	**VTE Communication error**	
Scan:	Cyclic, every time data is transmitted from the VT 340 to the computer	
Effect:	The RS232C (V.24) connection between the VT 340 and the computer has broken down.	
Explanation:	While data was being transmitted (VT 340 to computer), a fault occurred which is hindering transmission (e.g. connection cable not in position). Transmission will not be possible until the problem has been eliminated.	
Remedy:	Check the RS232C (V.24) communication program in the computer or check the RS232C (V.24) connection between the VT 340 and the computer.	

d) from GA2, Software Version 1

© Siemens AG 1991 All Rights Reserved

3150	**No reaction from PLC**	
Scan:	On function start via softkey	
Effect:	None	
Explanation:	• The alarm is displayed with the serial number • Alarm 3150 appears after approx. 0.5 to 2 seconds if the PLC to be cancelled by overall reset is not fitted or if a selected PLC fails during the overall reset operation	
Remedy:	• Overall reset of correct (existing) PLC • Replace PLC	

3151	**Interface occupied by PG**	
Scan:	On function start via softkey	
Effect:	None	
Explanation:	• The alarm is displayed with the serial number • Alarm 3150 appears if data are just being transferred between the programmer and one of the PLCs	
Remedy:	• Abort programming function • Terminate programming function	

3152	**Incorrect PLC mode**	
Scan:	On function start via softkey	
Effect:	None	
Explanation:	• The alarm is displayed with the serial number • Wrong operation mode set at the PLC	
Remedy:	Set correct operating mode (cycle/stop)	

3153	**Funct. ident. unknown in PLC**	
Scan:	On function start via softkey	
Effect:	None	
Explanation:	• The alarm is displayed with the serial number • The correct function identifier is unknown to the PLC	
Remedy:	Select correct function	

© Siemens AG 1991 All Rights Reserved

3154	Error in PLC program memory	
Scan:	On function start via softkey	
Effect:	None	
Explanation:	• The alarm is displayed with the serial number • Error in the PLC program memory	
Remedy:	Report to Service	

3155	PLC RAM not inserted	
Scan:	On function start via softkey	
Effect:	None	
Explanation:	• The alarm is displayed with the serial number • The PLC RAM is not inserted	
Remedy:	Insert PLC RAM	

3156	Function disabled or active	
Scan:	On function start via softkey	
Effect:	None	
Explanation:	• The alarm is displayed with the serial number • The PLC function is disabled because another funciton is active	
Remedy:	Avoid simultaneous signal status display at the programmer	

3157	Halt in thread	
Scan:	On thread cutting	
Effect:	• Processing interruption • NC START disabled	
Explanation:	• The alarm is displayed with the block number • During thread cutting a halt has occurred in revolutional feed which has damaged the thread	
Remedy:	Check axis-specific feed disable (DB 32)	

© Siemens AG 1991 All Rights Reserved

3158	PLC not available	
Scan:	On selection via softkey in a programmed display	
Effect:	None	
Explanation:	• The alarm is displayed with the serial number • The programmed field refers to a non-existent PLC number specification	
Remedy:	Check assignments using the programming workstation and correct	

3159	Data block not available	
Scan:	On selection via softkey in a programmed display	
Effect:	None	
Explanation:	• The alarm is displayed with the serial number • The programmed field refers to a non-existent data block	
Remedy:	Check assignment using the programming workstation and correct	

3160	Tool simul. w/o dimensions	
Scan:	While simulation is active	
Effect:	Simulation is stopped	
Explanation:	• Alarm is displayed with the block number • The programmed T number does not exist in the simulation tool data	
Remedy:	Check the programmed T number in the simulation tool data and enter new values if necessary	

3161	Tool form simulation not possible	
Scan:	While simulation is active	
Effect:	The simulation is stopped	
Explanation:	• The alarm is displayed with the block number • An illegal tool form is defined in the simulation tool data under the programmed T number	
Remedy:	• Define other tool form • Check program	

© Siemens AG 1991 All Rights Reserved

3162	**Not all programs deleted**	
Scan:	On reading-in via the serial interface	
Effect:	None	
Explanation:	• Alarm is displayed with the block number • An entire program section is to be deleted with %CLF on reading-in via the serial interface. For this, a program number has been specified which is just being processed and thus cannot be deleted	
Remedy:	Select ”Clear program” with %CLF via the serial interface once again after the program concerned has been completed	

3163	**Softkey start not possible**	
Scan:	On starting a subroutine via softkey	
Effect:	The subroutine is not started	
Explanation:	• The alarm is displayed with the serial number • The subroutine to be started in a defined channel cannot be started via softkey	
Remedy:	Check whether • Channel is available • Subroutine with @00f in 1st block • Channel is already assigned (program running)	

3164	**Axis converter error**	
Scan:	• On processing in AUTOMATIC or MDA mode • On graphic simulation	
Effect:	Processing interruption	
Explanation:	• Alarm is displayed with the channel number and the block number • Incorrect input in the axis conversion list (SD) – Axis name not entered – Axis name does not exist – Axis fictitious (for transformation)	
Remedy:	Correct axis conversion list	

© Siemens AG 1991 All Rights Reserved

3165[d]	**Remote PG has failed**	
Scan:	Cyclic, when remote PG has been selected	
Effect:	None	
Explanation:	The alarm is displayed with the serial number	
Remedy:	• Check CP231 • Check software version of CP231 • Delete CP231 dual port RAM (installation mode) if necessary	

3166	**Program coordination wrong**	
Scan:	On processing in AUTOMATIC mode	
Effect:	Machining is stopped in the event of grave errors by the PLC (Read-in disable)	
Explanation:	• Alarm is displayed with the channel number and the block number • The channel specified in the program coordination command WAIT or INIT does not exist • The channel specified in the program coordination command START does not exist or is disabled for processing	
Remedy:	• Check program coordination command in part program • Check all associated channels with a WAIT M command	

3167	**Reassign T/H word**	
Scan:	Each time the T/H word is rearranged	
Effect:	• Processing interruption • NC START disabled	
Explanation:	• The alarm is displayed with the serial number • The part program can be used to transfer T and H words from the NC channel in which they have been progammed (source channel) to another NC channel (target channel). Each reassignment must be acknowledged by the user in the target channel before the next reassignment is made • Possible errors: – Target channel not enabled via machine data (signal from/to channel) – Reassignment in the target channel has not yet been acknowledged by the user	
Remedy:	• Match part and user program • Enable target channel by machine data	

d) from GA2, Software Version 1

© Siemens AG 1991 All Rights Reserved

3168	**Coord. not reproducible**	
Scan:	On processing in AUTOMATIC mode	
Effect:	None	
Explanation:	• Alarm is displayed with the block number • Programming error in T/H word reassignment	
Remedy:	Match part and user program	

3170	**FA1 KUE wrong**	
3171	**FA2 KUE wrong**	
3172	**FA3 KUE wrong**	
Scan:	On calculating the transmission ratios for an ELG or a sychro-spindle pair	
Effect:	• Command ”Activate Kü” disabled (emergency signal or SD bit) • The old transmission ratio is maintained	
Explanation:	• The alarm is displayed with the serial number • An error has occurred in the calculation of the transmission ratio. Possible causes are: – Denominator= 0 – Result too large (> 1000) – Format overrun caused by unfavourable selection of increment weighting and axis-specific resolutions	
Remedy:	• Check transmission parameters • Check increment weighting and axis-specific resolutions	

© Siemens AG 1991 All Rights Reserved

3173	**FA1 Centering flank violated**	
3174	**FA2 Centering flank violated**	
3175	**FA3 Centering flank violated**	
Scan:	When the electronic gear (ELG) is active and when the emergency signal HAE IS ACTIVE is set	
Effect:	• NC START disabled • A movement beyond the already recognized flanks is suppressed	
Explanation:	• The alarm is displayed with the serial number • For semi-automatic centering, the two tooth flanks are approached manually. A movement beyond the already recognized flanks is suppressed to make centering easier for the operator. The alarm is also acitvated if the interface signals HAE ON and 1ST FLANK IS APPROACHED are set without the FA being traversed in between.	
Remedy:	• Withdraw FA from flank (opposite to approach direction) • Remove flank again if necessary (cancel interface signal 1ST FLANK IS APPROACHED) • Traverse at least one increment between the interface signals HAE ON and 1ST FLANK IS APPROACHED	

3176	**FA1 Synch. parameter wrong**	
3177	**FA2 Synch. parameter wrong**	
3178	**FA3 Synch. parameter wrong**	
Scan:	On change of synchronisation parameters	
Effect:	• NC START disabled • The last synchronisation parameters entered remain valid	
Explanation:	• The alarm is displayed with the serial number • The synchronisation parameters have been incorrectly specified. The following checks are made: – Has a valid value been entered for the number of the LA? The position of the leading axis is ignored with LA number 0. – Are the values within the permissible no. range (limits ±1070 000)?	
Remedy:	• Enter permissible LA number (0, 1 to 5) • Enter permissible synchronisation positions	

© Siemens AG 1991 All Rights Reserved

3200	Prog. coord. syntax wrong	
Scan:	On processing in AUTOMATIC or MDA mode	
Effect:	Processing interruption	
Explanation:	• The alarm is displayed with the channel number and the block number • Syntax error: invalid command mnemonics • Invalid modification parameter, impermissible characters, too many parameters, parameters or characters missing, range limit violation by parameter	
Remedy:	Correct wrong command	

3201	Prog. coord. too many parameters	
Scan:	On processing in AUTOMATIC or MDA mode	
Effect:	Processing interruption	
Explanation:	• The alarm is displayed with the channel number and the block number • More command parameters have been programmed than allowed in the command description	
Remedy:	Correct wrong command	

3202	Prog. coord. range limit violation	
Scan:	On processing in AUTOMATIC or MDA mode	
Effect:	Processing interruption	
Explanation:	• The alarm is displayed with the channel number and the block number • One of more range parameters violate the permissible values for the upper or lower limit	
Remedy:	Correct wrong command	

3203	Prog. coord. illegal character	
Scan:	On processing in AUTOMATIC or MDA mode	
Effect:	Processing interruption	
Explanation:	• The alarm is displayed with the channel number and the block number • Illegal characters occur in the coordination command	
Remedy:	Correct wrong command	

© Siemens AG 1991 All Rights Reserved

3204	**Prog. coord. command incomplete**	
Scan:	On processing in AUTOMATIC or MDA mode	
Effect:	Processing interruption	
Explanation:	• The alarm is displayed with the channel number and the block number • Parameter or right bracket missing in the programmed command	
Remedy:	Correct wrong command	

3205	**Prog. coord. R parameter error**	
Scan:	On processing in AUTOMATIC or MDA mode	
Effect:	Processing interruption	
Explanation:	• The alarm is displayed with the channel number and the block number • An error has occured on activation of programmed R parameters	
Remedy:	Correct wrong command	

3206	**Prog. coord. sym. para. not allowed**	
Scan:	On processing in AUTOMATIC or MDA mode	
Effect:	Processing interruption	
Explanation:	• The alarm is displayed with the channel number and the block number • No symbolic parameters are permitted except R parameters	
Remedy:	Correct wrong command	

3220	**Change G176 G175**	
Scan:	On processing in AUTOMATIC or MDA mode	
Effect:	The acitvated freezing function is aborted and a transition is made to the G function 175	
Explanation:	• The alarm is displayed with the channel number and the block number • The zero offset group or the D number has been changed, an angle of rotation offset has been activated or G53 has been programmed while the freezing function for angle of rotation, zero offsets or length compensations were active. These functions cause the ”Freezing” function to be terminated and the G175 function to be changed	
Remedy:	Check program blocks and correct	

© Siemens AG 1991 All Rights Reserved

3239	**Cycle exceeded by SPF**	
Scan:	On reading in subroutines via RS232C (V.24) or file transfer	
Effect:	Eprom cycle is overwritten by subroutine	
Explanation:	• The alarm is displayed with the channel number and the block number • A subroutine has been read in whose number already existed as Eprom cycle. The Eprom cycle is no more active.	
Remedy:	• The subroutine read in must be deleted if the Eprom cycle is to be maintained • If overwriting is to be prevented altogether, the setting data special bit 2, bit 3 of the relevant RS232C (V.24) interface or MD 5147, bit 3 in the case of file transfer must be set to 0	

3240	**SPF not read in**	
Scan:	On reading in subroutines via RS232C (V.24) or file transfer	
Effect:	Subroutine has not been read in	
Explanation:	• The alarm is displayed with the channel number and the block number • A subroutine to be read in already exists as Eprom cycle	
Remedy:	If the Eprom cycle is to be overwritten, the setting data special bit 2, bit 3 of the relevant RS232C (V.24) interface or MD 5147, bit 3 in the case of file transfer must be set to 1	

3241 d)	**M function programmed wrong**	
Scan:	On each block change	
Effect:	Processing interruption	
Explanation:	• The alarm is displayed with the channel number and the block number. • An incorrect M function has been programmed in the block: – Decade number has been exceeded – Minus sign in front of equals sign (e.g. M-1 = 3) – M02/M03 programmed as rapid auxiliary function	
Remedy:	Alter part program accordingly	

d) from GA2, Software Version 1

© Siemens AG 1991 All Rights Reserved

3242[d)]	**S function programmed wrong**	
Scan:	On each block change	
Effect:	Processing interruption	
Explanation:	• The alarm is displayed with the channel number and the block number. • An incorrect S function has been programmed in the block: – Decade number has been exceeded Dwell time > 99.9 revolutions Spindle position on oriented spindle stop > 360° Spindle value > 99999.9 – Minus sign in front of equals sign (e.g.: S-1 = 3) – Negative sign entered	
Remedy:	Alter part program accordingly	

3243[d)]	**T function programmed wrong**	
Scan:	On each block change	
Effect:	Processing interruption	
Explanation:	• The alarm is displayed with the channel number and the block number. • An incorrect T function has been programmed in the block: – Minus sign in front of equals sign (e.g.: T-1 = 3) – Negative sign entered	
Remedy:	Alter part program accordingly	

3244[d)]	**D function programmed wrong**	
Scan:	On each block change	
Effect:	• Processing interruption	
Explanation:	• The alarm is displayed with the channel number and the block number. • An incorrect D function has been programmed in the block: – Decade number has been exceeded – Minus sign in front of equals sign (e.g.: D-1 = 3) – Minus sign entered – Programmed D number does not lie in the TO area of the channel	
Remedy:	Alter part program accordingly	

d) from GA2, Software Version 1

© Siemens AG 1991 All Rights Reserved

3245[d]	**H function programmed wrong**	
Scan:	On each block change	
Effect:	• Processing interruption	
Explanation:	• The alarm is displayed with the channel number and the block number. • An incorrect H function has been programmed in the block: – Minus sign in front of equals sign (e.g.: H-1 = 3)	
Remedy:	Alter part program accordingly	

3246[d]	**F function programmed wrong**	
Scan:	On each block change	
Effect:	• Processing interruption	
Explanation:	• The alarm is displayed with the channel number and the block number. • An incorrect F function has been programmed in the block: – Minus sign in front of equals sign (e.g.: F-1 = 3) – Minus sign entered	
Remedy:	Alter part program accordingly	

3747	**No M19 with V-constant**	
Scan:	In the block or on each block change	
Effect:	The alarm is displayed with the block number • Processing interruption • NC START disabled	
Explanation:	M19 has been programmed with constant cutting speed (G96 S..).	
Remedy:	Deselect constant cutting speed with G97 before programming M19.	

4100	**No D number active**	
Explanation:	CRC/SRK have been selected without entering a D number in a standard cycle.	
Remedy:	Check part program	

d) from GA2, Software Version 1

© Siemens AG 1991 All Rights Reserved

4101	**Tool radius = 0**	
Explanation:	The cutter radius has been entered as 0. With standard cycles this causes an error.	
Remedy:	Enter radius in the D number	

4102	**Cutter radius too large**	
Explanation:	If this cutter is used, contour errors will result in several of the standard cycles.	
Remedy:	Program a different cutter	

4103	**Tool too wide**	
Explanation:	The width of the recessing tool is too great for the standard grooving cycle	

4120	**No spindle direction programmed**	
Explanation:	No spindle direction was programmed before the standard cycle was called.	
Remedy:	Alter part program	

4121	**Spindle not in tolerance range**	
Explanation:	In case the spindle speed fluctuates too much when a standard cycle is used.	
Remedy:	Check drive actuator, MDs	

© Siemens AG 1991 All Rights Reserved

4122[d]	**Calculated feedrate too large**
Explanation:	The feedrate calculated for the rotary axis in the cycle is larger than the maximum permissible feedrate of the axis.
Remedy:	• Check the speed programmed in the S word before calling the cycle and reduce it. • Check whether the correct maximum axis speed has been entered in the machine data.

4140	**Machined part diameter too small**	
Explanation:	The value for the finished part diameter defined in the standard cycle parameters is too small.	
Remedy:	Check part program	

4153	**Thread length too short**	
Explanation:	The overflow offset calculated in L84 is longer than the thread length.	
Remedy:	• Program a smaller spindle speed • Deselect overflow offset • Set point on reference plane higher	

4180	**Option not available**	
Explanation:	The standard cycle which has been called requires an option which is not available.	
Remedy:	Install option	

4200	**Check definition R (Nxxx)**	
Explanation:	The defining parameter has been defined incorrectly.	
Remedy:	Redefine parameter correctly	

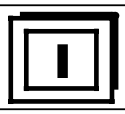
d) from GA2, Software Version 1

© Siemens AG 1991 All Rights Reserved

5000 : 5099	**User cycle alarms**	
Explanation:	The alarm has been called with @ 4c0 in the user cycle.	
Remedy:	Check the cycle	

6000 : 9999	**PLC error message**	
Explanation:	An initiation bit has been set in the PLC user program.	
Remedy:	According to machine tool manufacturer's instructions	

6000 : 9999	**PLC operational messages**	
Explanation:	An initiation bit has been set in the PLC user program.	
Remedy:	The operational message is acknowledged automatically when the initiation bit is reset.	

1000*	**Connection assigned more than once**	
Scan:	On POWER ON	
Effect:	• Processing interruption • NC START disabled • Opening of NC Ready Relay • Servo enable is removed after the time in NC MD 156 has elapsed (servo enable relay drops out) • Follow-up mode	
Explanation:	• The alarm is displayed with the serial number • A measuring circuit connection has been entered in NC MD 200* or 384* more than once • In addition, alarm 58 "Parameterzation error" is also triggered	
Remedy:	• Check NC MD 200* • Check NC MD 384*	

© Siemens AG 1991 All Rights Reserved

1004*	**Feedrate too high**	
Scan:	On each axis motion	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Servo enable removed	
Explanation:	• The alarm is displayed with the serial number • Excessive values have entered in MD 280* , i. e. the maximum value has been exceeded • The limiting frequency of the C axis encoder (MD 308*) has been exceeded with a spindle operated as rotary axis (C axis)	
Remedy:	• Check MD 280* and MD 155 • Check MD 308* • Reduce feedrate or speed	

1008*	**Speed controller limitation**	
Scan:	Cyclic	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready	
Explanation:	• The alarm is displayed with the serial number • NC has switched to follow-up mode and disables rpm servo control and current control • The speed controller has been at its limitation for longer than the time set in IAR MD 52* • Error in parameter assignment to the ACC module • Load moments of inertia too high • SIMODRIVE fault • Rotor position encoder signals to the ACC module interrupted • Speed alarm limit has been reached	
Remedy:	• Check parameter assignment to ACC module (IAR MD 136*; 140*; 144*; 148*) • Reduce load moments of inertia • Check SIMODRIVE • Check transmission of rotor position encoder signals to the ACC module • Set speed control loop to a slower speed • Increase parameter setting	

© Siemens AG 1991 All Rights Reserved

1012*	**Parameter assignment error** c) **Param. error drive MD** d)	
Scan:	• On warm restart • After change of MD	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready	
Explanation:	• The alarm is displayed with the serial number • NC switches to follow-up mode and disables rmp servo control and current control • Parameter assignment error in NC or IAR machine data	
Remedy:	• Check the following machine data for practical values: – IAR MD 108*c)/MD 1396*d) – IAR MD 116*c)/MD 1204*d) – IAR MD 128*c), 132*c)/MD 1208*d), MD 1212*d) – IAR MD 212*c) – IAR MD 268*c)/MD 1336*d) – NC MD 160 – NC MD 264*, 268* – NC MD 364*, 368* – NC MD 1116*/MD 1682*d)	

1016*d)	**MS switchover not poss.**	
Scan:	When switching over the measuring system	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Follow-up mode • Setpoint = 0	
Explanation:	• The alarm is displayed with the serial number • When a request is made to switch over the position measuring system, a check is made whether – a second measuring system is available – the actual value difference is smaller than the tolerance (MD 1216*) If one of the conditions has not been fulfilled, the alarm is triggered.	
Remedy:	• Introduce a second measuring system • Increase tolerance range (MD 1216*)	

c) up to and including Software Version 6

d) from GA2, Software Version 1

© Siemens AG 1991 All Rights Reserved

1024*c)	**Illegal pulse multiplication**	
Scan:	• On POWER ON • After change of MD	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay • Servo enable is removed after the time in NC MD 156 has elapsed (servo enable relay drops out) • Follow-up mode	
Explanation:	• The alarm is displayed with the serial number • A value other than 1, 2, 4, 8, 16, 32, 64 or 128 has been entered in NC MD 1116*	
Remedy:	Correct NC MD 1116*	

1036*c)	**Actuator not ready**	
Scan:	Cyclic	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready	
Explanation:	• The alarm is displayed with the serial number • NC switches to follow-up mode and disables rmp servo control and current control • Actuator not ready	
Remedy:	Check actuator (supply voltage)	

1040*	**Absolute encoder error**	
Scan:	On POWER ON	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Control does not start	
Explanation:	• The alarm is displayed with the serial number • An absolute encoder error is scanned with ”Power On”. The error cause appears in the service status display (see also Section ”Absolute encoder errors”).	
Remedy:	• Check connection measuring circuit-absolute encoder • Remove cause of error	
Caution:	The control must be restarted after the error has been remedied	

c) up to and including Software Version 6

© Siemens AG 1991 All Rights Reserved

1044*	**Battery absolute submodule**	
Scan:	Approx. every 10 minutes	
Effect:	Risk of losing absolute position on ”Power Off”	
Explanation:	• The alarm is displayed with the serial number • The battery in the absolute submodule is empty	
Remedy:	Replace battery under power	
Caution:	**The control must not be switched off until the battery has been replaced**	

1052*d)	**Drive fault**	
Scan:	Cyclic	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready	
Explanation:	• The alarm is displayed with the serial number. • Group error when drive fault occurs (SIMODRIVE), e.g.: local bus fault, computing time overflow, motor overtemperature, transistor fault, power section overheating, undervoltage/overvoltage, power section not ready, mains power fault, maximum capacity or continuous capacity exceeded	
Remedy:	• Check power supply of drive actuator • Check power consumption and heat removal of drive actuator • Check drive actuator hardware	

1060*d)	**NC command illegal**	
Scan:	Depending on operation	
Effect:	• Processing interruption • NC START disabled	
Explanation:	• The alarm is displayed with the serial number • This alarm is triggered when the NC has no control command authorization to the drive actuator (SIMODRIVE) but still wants to – change the operating mode (e.g. from open-loop control mode to axis mode) – define the setpoints (e.g: while the function generator is in operation)	
Remedy:	Enable authorization (via softkey)	

d) from GA2, Software Version 1

© Siemens AG 1991 All Rights Reserved

1200*	**Wrong division**	
Scan:	Cyclic	
Effect:	• Processing interruption • NC START disabled	
Explanation:	• The alarm is displayed with the serial number • The ”Division increment from PLC” function is not possible: – NC MD 1104* (number of divisions) has invalid value – NC MD 1108* (division increment reference) has invalid value – NC MD 1112* (division increment offset) has invalid value	
Remedy:	• Graduated axis has been defined as a rotary axis (NC MD 560* bit 3=1) which is not allowed • Check NC MD 1104*, 1108*, 1112* and 560* bit 3	

1204*	**Traversing range limit**	
Scan:	Cyclic (for linear axes only)	
Effect:	• Processing interruption • NC START disabled	
Explanation:	• The alarm is displayed with the serial number • On principle, it is possible to exceed the maximum possible traversing range (defined by the combination of axis-specific position control resolution and input resolution) if no software limit switches and working area limitations are active (NC MD 560* bit 5=0). Considering , however that this would lead to traversing errors, the traversing range limit is monitored and alarm 1204* is set when the limit is exceeded	
Remedy:	Return to permissible traversing range in INC or JOG mode traversing in opposite direction	

1208*	**G200 Axis in more than one channel**	
Scan:	When an axis is to be synchronized with G200 in one channel	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready	
Explanation:	• The alarm is displayed with the serial number and with the channel number • One axis is active also in other channels with G200	
Remedy:	Write part programs such that no G200 is requested with active axes	

© Siemens AG 1991 All Rights Reserved

1212*	**Position behind software limit switch**	
Scan:	On processing in AUTOMATIC or MDA mode	
Effect:	• Processing interruption • NC START disabled • Programmed path is **not** traversed	
Explanation:	• The alarm is displayed with the channel number and the block number • The programmed end point of the block is located behind the software limit switch	
Remedy:	• Correct program • Check MD 244*, 228*, 232*, 236*, dependent on the PLC interface signal 2ND SOFTWARE LIMIT SWITCH ACTIVE	

1216*	**Position behind working area limitation**	
Scan:	On processing in AUTOMATIC mode	
Effect:	• Processing interruption • NC START disabled • Programmed path is **not** traversed	
Explanation:	• The alarm is displayed with the channel number and the block number • The programmed end point of the block is located **behind** the working area limit	
Remedy:	• Correct program	

1232*	**Comp. limit switch +**	
Scan:	• Cyclic • When the axes are traversed • When the compensation values are not equal to 0	
Effect:	• Processing interruption • NC START disabled • Travel direction and compensation disabled.	
Explanation:	The effective machine position (NC setpoint position - compensation value) defined by the software limit switch (or working area limitation) has been reached. Traversing by the NC and compensation in this direction is disabled.	
Remedy:	Leave limit switch in JOG mode or move axis in the opposite direction by changing the compensation parameters	

© Siemens AG 1991 All Rights Reserved

1236*	**Comp. limit switch -**	
Scan:	• Cyclic • When the axes are traversed • When the compensation values are not equal to 0	
Effect:	• Processing interruption • NC START disabled • Travel direction and compensation disabled.	
Explanation:	The effective machine position (NC setpoint position - compensation value) defined by the software limit switch (or working area limitation) has been reached. Traversing by the NC and compensation in this direction is disabled.	
Remedy:	Leave limit switch in JOG mode or move axis in the opposite direction by changing the compensation parameters	

2000*d)	**Comp. grid spac. inval.**	
Scan:	Cyclic	
Effect:	• NC START disabled	
Explanation:	• The alarm is displayed with the serial number. • The permissible range (0 ... 1000 leadscrew error compensation points) was exceeded when the drive was operated. • For the leadscrew error compensation for the rotary axis, a value into which 360 (degrees) cannot be divided exactly has been entered in MD 324* for the axis concerned. The grid spacing is therefore not equal.	
Remedy:	Alter MD 324*	

2001*d)	**Rpm setpoint warn limit**	
Scan:	Cyclic	
Effect:	• NC START disabled	
Explanation:	• The alarm is displayed with the serial number • If the defined speed is too high, the drive vibrates • The digital speed setpoint in the drive unit has exceeded the warning limit	
Remedy:	• Enter a lower speed • Check actual values (pulse encoder)	

d) from GA2, Software Version 1

© Siemens AG 1991 All Rights Reserved

2003*d)	**Zero speed monitoring** 
Scan:	• On clamping • On standstill • On uneven deceleration • In synchronous spindle operation
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay (after the time defined in MD 156)
Explanation:	• The alarm is displayed with the serial number • The following error could not be cleared faster than the time entered in NC MD 156. • On clamping, the limit defined in NC MD 212* was exceeded. • A mechanically clamped axis has been pushed out of position. • Fault in: – control device (actuator) – tachogenerator – motor – NC measuring circuit hardware – pulse encoder • Wrong position feedback polarity on installation.
Remedy:	• NC MD 212* (clamping tolerance) must be greater than MD 204* (exact stop limit) • MD 156 (servo enable cutoff delay) must be large enough to allow the following error to be cleared within this period (only if MD 372* = 0). • MD 372* (zero speed control delay) must be large enough to allow the following error of the individual drives to be cleared within the defined period. • Check actual values (pulse encoder) and position control polarity

2007*d)	**Meas. loop not available** 
Scan:	On POWER ON
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready
Explanation:	• The alarm is displayed with the serial number • Measuring circuits have been assigned to axes or spindles in NC MD 200* or 400* which have been recognized by the NC as non-existent after POWER ON. • The measuring circuit module has either been removed or is defective.
Remedy:	• Check machine data • Check measuring circuit module

d) from GA2, Software Version 1

© Siemens AG 1991 All Rights Reserved

2008*d)	**Control loop hardware spindle**	
Scan:	Cyclic	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay (after the time defined in MD 337*)	
Explanation:	• The alarm is displayed with the serial number • The measuring circuit differential signals: – are not in phase – have a short circuit to earth – do not exist at all	
Remedy:	• Check that the measuring circuit connector is inserted • Check the measuring circuit module (plug in measuring circuit short-circuit connector) • Check differential signals with oscilloscope • Replace measuring sensors/cable	

2009*d)	**Dirt in measur. system**	
Scan:	Cyclic	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready	
Explanation:	• The alarm is displayed with the serial number • In measuring systems with contamination signal, the measuring system signals an error to the NC.	
Remedy:	• Check measuring system	

2011*d)	**Zero mark monitoring**	
Scan:	Cyclic	
Effect:	• Processing interruption • NC START disabled	
Explanation:	• The alarm is displayed with the serial number • Pulses have been lost as a result of transmission errors or interference, with reference to one encoder revolution.	
Remedy:	• Check encoder pulses • Check transmission path • Switch off monitoring temporarily (hardware monitoring off)	

d) from GA2, Software Version 1

© Siemens AG 1991 All Rights Reserved

2014*d)	**Rpm setpoint alarm lim.** 
Scan:	Cyclic
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Setpoint 0 • The servo enable is removed after the time stored in NC MD "Servo enable cutout delay" • Follow-up mode
Explanation:	• The motor cannot follow the speed command value • On installation: wrong position feedback polarity, wrong spindle multgain. The actual speed value has exceeded the maximum spindle speed + tolerance.
Remedy:	• Match spindle multgain (MD 468*) • Increase the tolerance of the maximum spindle speed (MD 445*) • Increase the acceleration time constant (MD 419* - 426*) • Check the drive • Check position feedback polarity • Check speed setpoint cable • Check actual value (pulse encoder)

2015*d)	**Spindle drift too high**
Scan:	When NC MD 401* is entered
Effect:	NC START disabled
Explanation:	The entered drift value is greater than 500.
Remedy:	Check whether the drift on the drive unit has been correctly adjusted.

d) from GA2, Software Version 1

© Siemens AG 1991 All Rights Reserved

2016*d)	**Conn. assign. 2x or more**	
Scan:	After POWER ON	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Opening of NC Ready Relay (after the time defined in MD 156) • Follow-up mode	
Explanation:	• The alarm is displayed with the serial number. • A measuring circuit connection has been entered more than once in MD 400* or 460*.	
Remedy:	• Check MD 400* • Check MD 460*	

2018*d)	**Rpm controller limitat.**	
Scan:	Cyclic	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready	
Explanation:	• The alarm is displayed with the serial number • The NC has switched to follow-up mode and disabled speed and current control • The external moment of inertia is too high • A fault has occurred in the SIMODRIVE	
Remedy:	• Decrease external moment of inertia • Check SIMODRIVE	

d) from GA2, Software Version 1

© Siemens AG 1991 All Rights Reserved

2019*d)	**Param. error drive MD**	
Scan:	• During ramp-up • On warm restart • On MD changes	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready • Display of a service number in the diagnostics display for the spindle concerned	
Explanation:	• The alarm is displayed with the serial number. • Errors were discovered when the MDs for the spindle or corresponding C axis were checked.	
Remedy:	Check the following MDs: 155, 160...163, 168, 200*, 364*, 368*, 384*, 400*, 403*...410*, 419*...426*, 435*, 455*, 456*, 458*, 459*, 460*, 461*, 463*, 466*, 468*, 478*...485*, 1800*	

2020*d)	**M19-Posit. not in range**	
Scan:	When selecting M19	
Effect:	• Processing interruption • NC START disabled	
Explanation:	• The alarm is displayed with the serial number • The alarm is output if M19 has been programmed without an S value in MDA or in a part program and an illegal value has been entered in the spindle setting data for oriented spindle stop.	
Remedy:	Enter a permissible value in the setting data Range: 0 ... 359.99	

2028*d)	**MD M19 not selected**	
Scan:	• On processing programs in AUTOMATIC or MDA mode or when processing external programs is specified	
Effect:	Processing interruption	
Explanation:	• The alarm is displayed with the serial number • "M19 S.." has been triggered by the PLC even though this function has not been enabled.	
Remedy:	• Check part program or PLC user program • Check MD • Ask service to check function expansion and have it installed if necessary	

d) from GA2, Software Version 1

© Siemens AG 1991 All Rights Reserved

2029*d)	**Drive fault**	
Scan:	Cyclic	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready	
Explanation:	• The alarm is displayed with the serial number • Group error when drive fault occurs (SIMODRIVE), e.g.: local bus fault, computing time overflow, main spindle not ready, motor overtemperature, transistor fault, power section overheating, undervoltage/overvoltage, power section not ready, mains power fault, maximum capacity or continuous capacity exceeded	
Remedy:	• Check power supply of drive actuator • Check power consumption and heat removal of drive actuator • Check drive actuator hardware	

2030*d)	**Spindle speed too high**	
Scan:	Only when MD 520* (pulse encoder available) is set	
Effect:	• Processing interruption • NC START disabled • Cancellation of Mode Group Ready	
Explanation:	• The alarm is displayed with the serial number • The spindle speed has exceeded the value set in the machine data or the setting data.	
Remedy:	• Program a smaller S value • Check the following MDs: 403*...410*, 445*, 451* • Check PLC gear stage • Check "G92S.." at "v = constant" • Check "Spindle speed limitation" setting data	

d) from GA2, Software Version 1

© Siemens AG 1991 All Rights Reserved

2031*d)	**NC command illegal**	
Scan:	• When M19 is selected • Depending on type of operation	
Effect:	• Processing interruption • NC START disabled	
Explanation:	• The alarm is displayed with the serial number • The NC has accessed the drive unit even though it has no control command authorization (e.g output of M19 for spindle in C axis or synchronous mode).	
Remedy:	• Check part program • Check PLC program (M19 from PLC)	

d) from GA2, Software Version 1

© Siemens AG 1991 All Rights Reserved

4.1.8 Absolute encoder error

With the SIPOS absolute encoder function both errors in the encoder and on the separately arranged absolute submodule may occur.

It should be noted that a faulty connection between the encoder and the absolute submodule may activate almost all error messages because communication takes place via this connection. This transmission link must therefore be checked first when an error occurs.

Below all possible error numbers and error sources are listed, assuming a faultless connection between the encoder and the absolute submodule:

Error no.:	1 ... 50, 93 ... 195, 240 ... 253
Meaning: Remedy:	Faulty absolute encoder and**/**or absolute submodule • Check absolute encoder and absolute submodule for damage • Check connectors for fixed location
Error no.:	51 ... 70
Meaning: Remedy:	Absolute submodule and**/**or faulty connection between absolute submodule and measuring circuit • Check absolute submodule for damage • Check for fixed location of connector between measuring circuit and absolute submodule
Error no.:	71 ... 80, 200 ... 230
Meaning: Remedy:	Faulty absolute submodule**/**battery discharged (”Battery alarm” appears after 10 minutes) • Check absolute submodule for damage • Check encoder connector for fixed location
Error no.:	81 ... 86
Meaning: Remedy:	Faulty connection between absolute submodule and measuring circuit • Check absolute submodule for damage • Check for fixed location of connector between measuring circuit and absolute submodule

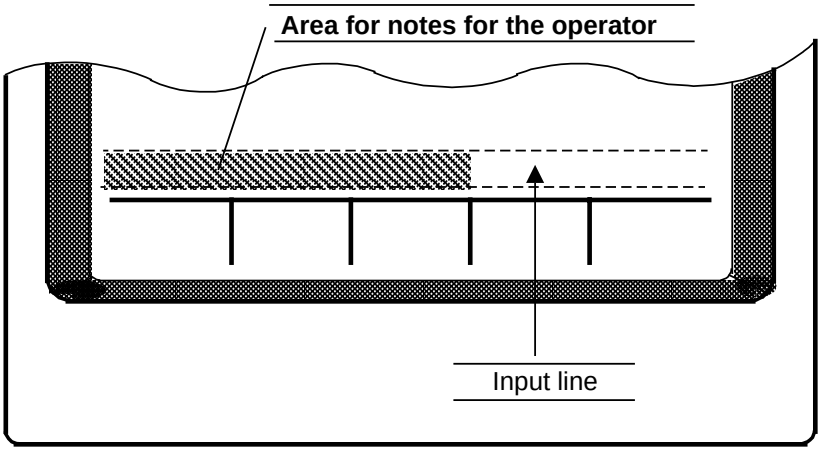
Note:

Do not fail to state the error number when sending a faulty component to be repaired.

© Siemens AG 1991 All Rights Reserved

4.2 Notes for the operator

With most operator inputs, texts are displayed in the left two thirds of the input line as support and for error diagnostics.



4.2.1 Listing of interactive dialog texts

All notes for the operator are listed in the following table in alphabetical order.

Interactive dialog text	Appears with the following event	What must be done? or What is wrong?
BLOCK DOES NOT EXIST	When searching for blocks in the part program	Select different block number
BLOCK NUMBER TOO LARGE	Generate block number via operator prompting	Block number may be max. N9995
BLOCK STRUCTURE INCORRECT	"Correction block" function	Cursor is at beginning of wrong block.
BLOCK TOO LONG	When editing the part program, a block must not be longer than 120 characters	Complete block with "CR"
CHARACTER DOES NOT EXIST	When searching for characters in part programs	--
CHARACTER NOT ALLOWED	Taking over an input value (e.g. with the input key)	Input value has the wrong format
CHECK REMOTE PG	Connection to remote PG has not been terminated	Check and if necessary reinitialize COM.CPU and CP231

© Siemens AG 1991 All Rights Reserved

Interactive dialog text	Appears with the following event	What must be done? or What is wrong?
COMMENT ERROR	Editing if bracket balance is incorrect	Check number of brackets set.
CONN. HAS BEEN CLEARED	After the connection has been successfully terminated	–
CONN. IS ACTIVE	After the connection has been successfully terminated	–
CONTROL BUSY	When the NC is overloaded	The command has been accepted but cannot be executed at present
CYCLE IN EPROM	Note when handling cycles	--
CYCLE LOCK	When handling CYCLES (EDIT, COPY, etc.)	Remove "Cycle lock" interface signal
DATA ERROR IN DRIFT COMPENSATION	With semiautomatic drift compensation	Repeat drift compensation
DATA TRANSMISSION ERROR	Data transmission via RS232C (V.24)	Transmission has been aborted
DATA TRANSMISSION RUNNING	When editing part programs	Stop interface
DECIMAL POINT NOT ALLOWED	Taking over an input value (e.g. with the input key)	--
DELETE DATA?	Acknowledgement key pressed	The specified data area is deleted when the key is pressed a second time
DIFFERENT PROGRAM TYPES	When copying or renaming part programs (COPY, RENAME)	Only identical program types can be handled (L to L, % to %, etc.)
DISPLAY DESCRIPTION	When selecting a configured display	Display description is faulty
EDITING NOT ALLOWED	When editing part programs	Repeat input
ENTER PASSWORD	When trying to enter protected data or after pressing diagnostic key	--
FORMAT ERROR	Taking over an input value (e.g. with the input key)	The control can do nothing with the value in this position. See User's Guide.

© Siemens AG 1991 All Rights Reserved

Interactive dialog text	Appears with the following event	What must be done? or What is wrong?
GENERAL DATA ERROR	Taking over an input value (e.g. with the input key)	The value cannot be taken over. Taking over the value would cause an error: change value.
INPUT DISABLED	Taking over an input value (e.g. with the input key)	Operate key-switch
ILLEGAL CHANNEL NUMBER	When copying tool data blocks (simulation)	Simulation channel not present
ILLEGAL LABEL	”Shopfloor programming” function. The label is incorrectly specified.	Only a two-digit numeral is permitted.
ILLEGAL NUMBER	Copying tool data blocks (simulation)	Invalid source or target number
ILLEGAL TOOL DATA	• Selecting simulation • Starting simulation**/**search simulation • Deselecting blank	• Min. Max. values • Working area limitations exceeded
ILLEGAL WORKING AREA	• Selecting simulation • Starting simulation**/**search simulation • Selecting window data • Inputting NC channel number (simul. data)	Working area machine data: Min. Max. values
INPUT ERROR (PROG:)	When editing part programs	The character selected cannot be entered in the part program.
INPUT LINE OVERFLOW	When entering characters	Max. 17 symbols possible
INTERFACE BUSY	When starting data input**/**output	Interface selected is already busy
INTERRUPT PROGRAM	Editing the program being processed	Interrupt program (NC STOP)
JOG MEASURING ABORTED	Error in JOG measuring or mode change	The function has been aborted.
KEY NOT ALLOWED	Taking over an input value (e.g. with the input key)	Refers to input and delete keys
KEY SWITCH MISSING	Selection of machine data tree or overall reset	Switch on key switch.

© Siemens AG 1991 All Rights Reserved

Interactive dialog text	Appears with the following event	What must be done? or What is wrong?
LABEL NOT DELETED IN PART PROGRAM	”Shopfloor programming” function	Part program is in EPROM or part program driver could not delete label.
MDA CHANNEL BUSY	MDA has already been started in other channel	Change channel or press RESET key.
MDA MEMORY OVERFLOW	In "Manual data input" mode	Max. 256 characters may be entered
MD TRANSFER ERROR	Fetching machine data (DAD error)	--
MEASURING ACTIVE!	JOG measuring if measuring operation has been activated.	--
MEMORY OVERFLOW	When entering part programs manually or via RS232C (V.24)	Part programs not to be used have to be deleted
"MENU TREE" NOT CONFIGURED	When paging through menu tree	Menu block/tree wrongly configured
"MOVE" PROGRAM	When editing part programs simultaneously with processing	Part program handling, MOVE or stop current program
NO CORRECTION BLOCK	With "Correction block" function	Correction block cannot be selected
NO LABEL	”Shopfloor programming” function	Part program does not contain this label.
NO PROGRAM NUMBER FREE	When opening a part program (see machine data 10)	The max. number of part programs has been reached; delete part programs
NO RENAME FOR CYCLES	Renaming cycles is not possible	--
NO ROTARY AXIS ENABLED	Switchover to rotary axis operation "Tapping without compensating chuck"	Initiate rotary axis operation (PLC)
NORMAL DISPLAY NOT PERMITTED	• Selecting simulation • Deselecting simulation data • Selecting normal display	• Min. Max. values • Working area limitations exceeded

© Siemens AG 1991 All Rights Reserved

Interactive dialog text	Appears with the following event	What must be done? or What is wrong?
NOT ALLOWED FOR SUB-ROUTINE	Block search in subroutine	Block search not allowed in subroutine. Delete specified block number in subroutine level.
NO VALUE ENTERED	Taking over an input value (e.g. with the input key)	Input value has the wrong format
ONLY 1 PROGRAM NUMBER ALLOWED	With MOVE or MACHINING YES/NO of a part program	--
ONLY POSSIBLE IN AUTOMATIC MODE	Selecting Teach in/ Playback	Switch to Automatic mode
ONLY POSSIBLE IN RESET MODE	When entering program numbers in the automatic basic display	The NC must be in the RESET state for safety reasons. Press RESET or wait until the current program is completed.
"OPERATOR PROMPTING FORM" BLOCK TOO LONG	When editing using screen forms with the aid of operat. prompting macros	The operator prompting screen form has been wrongly configured
"OPERATOR PROMPTING FORM" SEVERAL VALUES	When editing using screen forms with the aid of operator prompting macros	If several fields are bracketed in the display, only one of the values may be entered.
"OPERATOR PROMPTING FORM" VALUE MISSING	When editing using screen forms with the aid of operat. prompting macros	A value has not been entered
OPTION NOT FITTED	When executing a part program	The programmed function is not defaulted but available as an option.
OTHER INTERFACE ACTIVE	When starting data input/output	The other interface in the pair is already in operation
OVERSTORE ACTIVE	When overstoring	Wait until previous overstore operation has been completed
PRESELECT PROGRAM	Something is to be entered in a part program	Preselect part program
PROGRAM ALREADY EXISTS	When opening part programs (PP-SELECT, COPY, RENAME)	Select different number
PROGRAM DOES NOT EXIST	Taking over a part program number during part program handling	Correct number

© Siemens AG 1991 All Rights Reserved

Interactive dialog text	Appears with the following event	What must be done? or What is wrong?
PROGRAM ERASE-PROTECTED?	• Renaming part program number • Opening part programs • Reading in via RS232C (V.24) if part program is already available and is executed	Conclude execution or reading in
PROGRAM NUMBER NOT ALLOWED	Part program handling	Change part program number
PROGRAM PRESELECTION NOT ALLOWED	Preselection of part program during Teach in**/**Playback	Abort Teach in**/**Playback (RESET)
REMOTE PG IS INACTIVE	After connection has been successfully terminated	–
REMOTE PG RUNNING**/**STOP?	When remote PG is active and DATA ON-OFF and INPUT START softkeys have been pressed	–
REORG NOT POSSIBLE	When reorganizing; can only occur when the memory has been edited manually and is full	Delete blocks; if this does not help, part programs must be deleted
RETRACT PROBE	JOG measuring if the measuring operation has been concluded	Retract probe.
SAME PROGRAM NUMBERS	When copying or renaming part programs (COPY, RENAME)	The program numbers entered must be different
SEGMENT OVERFLOW	When editing manually	Reorganize (press REORG key)
SELECT JOG MODE	If JOG measuring is called in an operating mode other than JOG	Select JOG mode
SIMULATION ACTIVE	• Starting simulation**/** search simulation • Selecting window data • Selecting tool data • Inputting the NC channel number (simul. data)	• Starting simulation**/** search simulation is running • With search, simulation is stopped • When selecting window data**/**tool data and inputting channel number simulation is not in reset state

© Siemens AG 1991 All Rights Reserved

Interactive dialog text	Appears with the following event	What must be done? or What is wrong?
SIMULATION RUNNING	Repeated starting of simulation, although simulation program has not yet been terminated or aborted	Terminate or abort simulation program; restart simulation.
SOURCE PROGRAM ERROR	Part program read out operation (reading, copying ...)	A bit has overturned in the part program read (parity error).
SOURCE=TARGET (SIM. CHANNEL)	Copying tool data blocks (simulation)	Wrong input format. The channel number does not have to be specified.
SPECIFY 2ND PROGRAM NUMBER	When copying or renaming part programs (COPY, RENAME)	--
STOP AXES	The actual axis value is to be read (e.g. with PLAYBACK)	Axis values must not change.
TELE-S. BEING ACTIVATED	Service control has activated teleservice	–
TELE-S. BEING ESTABLISHED	Customer control establishing connection to service control	–
TELE-S. DISPLAY ACTIVE	Service control has activated teleservice	–
TELES-S. DISPLAY READY	Customer control has enabled teleservice; service cannot carry out any operations yet	–
TELE-S. ENDED	Teleservice has ended; both controls can again be operated locally	–
TELE-S. INPUT ACTIVE	Service control has activated teleservice	–
TELE-S. INPUT READY	Customer control has enabled teleservice; service carry out operations.	–
TOOL NUMBER NOT ALLOWED	In functions operating with tool offsets	e.g. PRESET with a rotary axis: In the PRESET display "0" must be entered for the tool number

© Siemens AG 1991 All Rights Reserved

Interactive dialog text	Appears with the following event	What must be done? or What is wrong?
TOO MANY CHARACTERS ENTERED	Taking over an input value (e.g. with the input key)	Input value has the wrong format
TRANSFER LABEL TO PART PROGRAM	”Shopfloor programming” function	Part program is in EPROM or is write-protected.
VALUE INCORRECTLY ENTERED	When inputting into display screen forms (e.g. contour definition)	Change value
WRONG AXIS SPECIFIED	With axis-specific functions (Preset, Playback ...)	Axis does not exist
WRONG AXIS TYPE	With axis-specific functions (Preset, Playback ...)	The function cannot be executed with this type of axis (e.g. automatic TO acquisition with rotary axis)
WRONG DATA NUMBER	When deleting data	Data item (e.g. R parameter) with the number entered does not exist
WRONG G FUNCTION	With technology-dependent functions	e.g. "Playback" will not function if G91 has been selected (NC can only store absolute values)
WRONG MODE GROUP	With functions assigned to a mode group (e.g. channel selection)	Select different mode group
WRONG PASSWORD	Password input	--
WRONG RS232C (V.24) PAIR	Starting or stopping data input/output	Select one of the correct interface pairs in the higher-level menu
WRONG RS232C (V.24) CHANNEL NUMBER	Starting or stopping data input/output	With 850/880: 1 to 6 possible
WRONG TOOL TYPE	With technology-specific functions (e.g. automatic TO acquisition)	Modify type if necessary in "Tool offset" display
1ST NUMBER > 2ND NUMBER	Deleting data areas	The first number must be smaller than the second
2ND DECIMAL POINT	Taking over an input value (e.g. with the input key)	Input value has the wrong format
"-" NOT ALLOWED	Taking over an input value (e.g. with the input key)	Input value has the wrong format

© Siemens AG 1991 All Rights Reserved

Interactive dialog text	Appears with the following event	What must be done? or What is wrong?
"=","CR" WRONG POSITION	Taking over an input value (e.g. with the input key)	Input value has the wrong format

© Siemens AG 1991 All Rights Reserved

5 Maintenance

5.1 Operating data

Humidity content is in accordance with DIN 40040	F
Air pressure	860 to 1080 hPa
Shockproof,type of protection acc. to DIN VDE 0160	I
Type of protection acc. to DIN 40050 • Front of operator panel • Rear of operator panel • Front of machine control panel • Rear of machine control panel	 IP 54 IP 00 IP 54 IP 00

Please refer to the Instruction Manual for complete data in use and operating data.

5.2 Replacing the battery

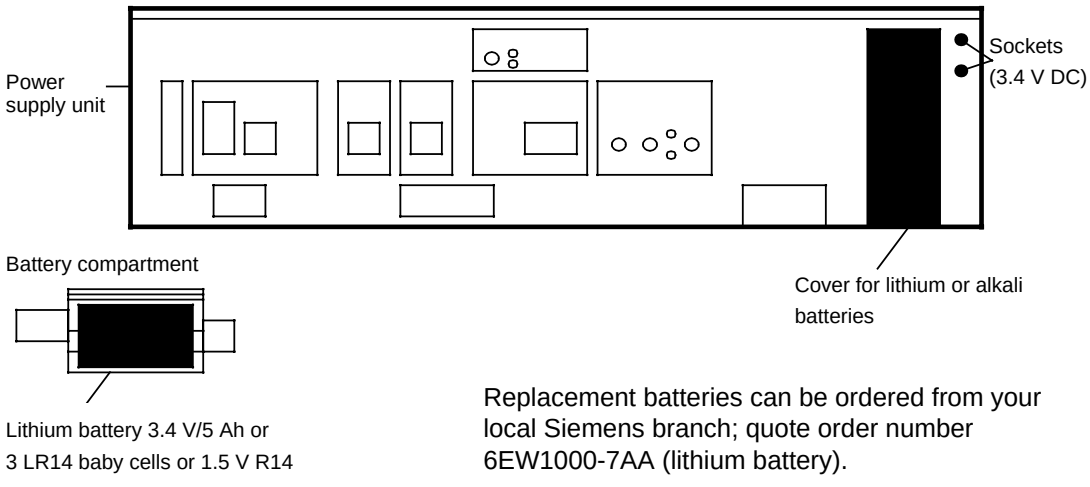
The backup battery voltage is monitored by the control.

If the battery voltage falls below 2.8 V, alarm 1 is triggered (Battery alarm power supply).

Once the monitoring has responded, the battery must be replaced within one week.

The battery may only be replaced if the control is switched on or a standby voltage (3.4 V DC) is applied at the sockets.

Please ensure that the polarity is correct when replacing the battery (red = +3.4 V; black = 0 V).
To replace the battery, push the cover down and remove the battery compartment.



Replacement batteries can be ordered from your local Siemens branch; quote order number 6EW1000-7AA (lithium battery).

© Siemens AG 1991 All Rights Reserved

Battery type:

Until the end of 1990:
3.4 V/5 Ah
TL 2200
IEC-R-14 (baby cell)
guaranteed lifetime 1 year

From the beginning of 1991:
1.5 V (baby)

A lithium battery which is new or has not been used for a relatively long time (> 5 weeks) builds up a deadening layer. This layer has the effect of increasing the internal resistance of the battery. To remove it, each new battery should be placed under load for 10 minutes at a resistance of 30 to 50 before being inserted in the control. Batteries already built into the control do not have tho be activated further because the deadening layer breaks down after a few days as a result of the basic load.

As special waste, old batteries must be disposed of separately from other refuse.

5.3 Memory submodule with back-up battery

The battery on the memory submodule backs up the machine data memory.

- The back-up time of the battery is 5 years.
- When alarm number "6" is displayed, the battery should be replaced.
- **The complete memory submodule must be replaced** after the battery life has expired.
- **The battery** on the memory submodule can only be exchanged in the manufacturer's works.

As from approx. 9/89 a machine data submodule is being introduced which no longer has a soldered in battery but which receives its power supply from the central backup battery (next to power supply unit).

Before replacing the submodule, the stored data must be saved. The memory submodule should be replaced by "Service". Replacement: the memory submodule with battery is insulated underneath. However, the module should never be laid on a conductive surface.

© Siemens AG 1991 All Rights Reserved

5.4 Handling modules

Nearly all the SINUMERIK modules have LSI chips. From a point of view of their technology, these are very sensitive to overvoltage and thus also to electrostatic discharge.

Modules using these components are identified on the component side by this warning symbol.



ELECTRONICALLY ENDANGERED COMPONENTS can be destroyed by voltages and energy levels well below those perceived by the human body.

Such voltages are present even if a component or module is touched by a person who has not undergone electrostatic discharge.

Components exposed to such overvoltages are not immediately recognizable as defective in most cases since faulty behauviour sets in only after they have been operating for a relatively long time.

Therefore, please pay attention to the following guidelines:

- Electronics modules should not be touched unless it is absolutely vital for work to be done.
- Before touching a module, any static in the body must be discharged. The easiest way to do this is by touching a conductive, earthed part of the structure (e.g. bare metal parts of the cabinet, socket earth contact).
- Modules should not be brought into contact with materials that are good insulators (e.g. plastic foil, insulated desk tops, clothing made from synthetic fibres).
- Modules should only be laid on conductive surfaces.
- Modules should only be plugged in or unplugged with the power off.
- The power supply should be switched on before any signal voltages are applied.
- Modules and components should always be stored and carried in conductive packing (e.g. metallised plastic boxes, metal containers).
- If the packing is not conductive, the modules should be wrapped in conductive material prior to packing. For example, conductive foam rubber or aluminium foil can be used.

© Siemens AG 1991 All Rights Reserved

5.5 Cleaning

The front of the monitor and the surface of the operating panel can be cleaned.

For dirt that is relatively easy to remove, standard household washing-up liquid, or an industrial cleaner (such as "Spezial Swipe") can be used.

These cleaners will also remove dirt containing graphite.

Cleaning agents which contain one or more of the following ingredients can be used for short periods of time:

- diluted mineral acids
- bases
- alcohol
- organic hydrocarbons
- detergent solutions

The plastic materials used on the front of the SINUMERIK 880 are suitable for use on machine tools. They are therefore resistant to

- greases and oils, mineral oils,
- bases and lyes,
- detergent solutions,
- alcohol.

Solvents such as chlorinated hydrocarbons, benzene, esters and ethers should be avoided.

© Siemens AG 1991 All Rights Reserved

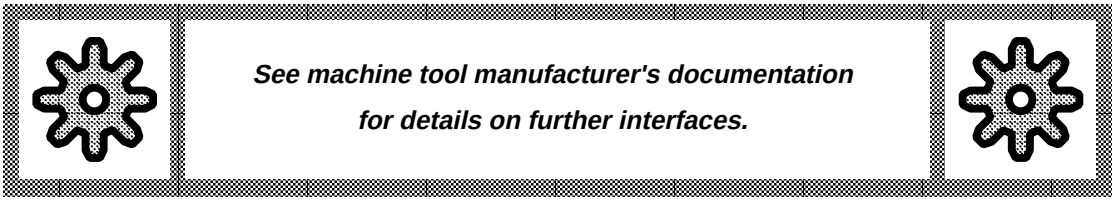
6 Data Interfaces

6.1 General

The Basic Version has two universal interfaces (RS232C (V.24)**/**20 mA) available for data transfer:

- **Interface 1** is variable with regard to the possibilities for connecting different devices**.**
- **Interface 3** is normally used for the connection of a particular device with fixed assignment.

Interfaces 2 and 4 are optional.



The interface data (Interface "characteristics") can be set separately via Setting Data (SD).

6.2 Setting data for description of the interfaces

On the SINUMERIK 880, the interfaces are described using setting Data (SD).

The interfaces 1 to 4 can be defined individually:

- **Interface 1:** SD No. **5010** to **5017 and 5026** to **5029**
- **Interface 2:** SD No. **5018** to **5025**
  (Option)
- **Interface 3:** SD No. **5030** to **5037 and 5046** to **5049**
- **Interface 4:** SD No. **5038** to **5045**
  (Option)

The EIA code for special characters is the same for both interfaces 1 and 2 and for 3 and 4 (SD numbers 5026 to 5029 and 5046 to 5049).

The following table contains a description of interfaces on the basis of Interface 1. All other interfaces can be described analogously with the respective SD numbers (see above).

© Siemens AG 1991 All Rights Reserved

Setting data function for interfaces 1/3

SD No.	Bit No. 7	6	5	4	3	2	1	0
5010 **5030**	Device coding for "Input" to RS232C (V.24)							
5011 **5031**	Transfer format for "Input" from RS232C (V.24) Number of stop bits		odd parity	with parity	baud rate			
	Stop bits: Number 00 = 1 01 = 1 10 = 1 1/2 11 = 2				baud rate: 0000 = 110 baud, 0001 = 150 baud, 0010 = 300 baud, 0011 = 600 baud, 0100 = 1200 baud 0101 = 2400 baud 0110 = 4800 baud 0111 = 9600 baud			
5012 **5032**	Device coding for "Output" to RS232C (V.24)							
5013 **5033**	Transfer format for "Output" from RS232C (V.24) Number of stop bits		odd parity	with parity	baud rate			
	Stop bits: Number 00 = 1 01 = 1 10 = 1 1/2 11 = 2				baud rate: 0000 = 110 baud, 0001 = 150 baud, 0010 = 300 baud, 0011 = 600 baud, 0100 = 1200 baud 0101 = 2400 baud 0110 = 4800 baud 0111 = 9600 baud			
5014 **5034**	XON start characters (RS232C (V.24))							
5015 **5035**	XOFF start characters (RS232C (V.24))							
5016 **5036**	Special bits RS232C (V.24) Start without X_{ON}	Progr. Start with LF	End of Block with CR LF	Output in EIA code	Stop at End of Transfer character	Data Set Ready is evaluated	Output without leading and trailing tape	Input program from System 3/8
5017 **5037**	Special bits RS232C (V.24)						No REORG via interfaces	Time out monitoring OFF

Description of the interfaces

© Siemens AG 1991 All Rights Reserved

SD No.	Bit No. 7	6	5	4	3	2	1	0
5026 5046	EIA code for "@" character							
5027 5047	EIA code for ":" character							
5028 5048	EIA code for "End of transfer" character							
5029 5049	EIA code for "=" character							

Description of the interfaces (continued)

© Siemens AG 1991 All Rights Reserved

6.3 Assignment of the setting data for the connection of peripheral devices

Before you connect a particular device to an interface, you must assign the "setting data bits" of the setting data number.

For that purpose, you call up the relevant setting data number (SD no.) – as described in Section 3.2.14.1.

As an example the SD numbers for Interfaces 1 and 3 are given.

Only the setting data numbers (SD no.) required for the connection of the individual devices are shown.

Peripheral device	Ident. of the device type (CRT display)	SD no. Interface 1	SD no. Interface 3	Required setting data bits (binary code)
SINUMERIK Reader: T40, T50,T60 RS232C (V.24), 9600 baud	RTS - LINE	5010 5011	5030 5031	0000 0000 1100 0111
SINUMERIK WS800 RS232C (V.24), 20 mA 9600 baud	RTS - LINE	5010 5011 5012 5013	5030 5031 5032 5033	0000 0000 1100 0111 0000 0000 1100 0111
Siemens Programmer unit PG 685/675/670 20 mA, 9600 baud Interface S5 PLC	PLC - PROG.	-------- --------	5030 5031	0000 0100 0000 0111
Siemens Programmer unit PG675 RS232C (V.24), 1200 baud Interface printer	RTS - LINE	5010 5011 5012 5013 5016 5028	5030 5031 5032 5033 5036 5048	0000 0000 1100 0100 0000 0000 1100 0100 0010 1000 0000 0011
Siemens Programmer unit PG 615 U, 20 mA, 9600 baud	PLC - PROG.	-------- --------	5030 5031	0000 0100 0000 0111
Siemens page printer PT 80 20 mA, RS232C (V.24), 300 baud	RTS - LINE	5010 5011 5012 5013	5030 5031 5032 5033	0000 0000 1100 0010 0000 0000 1100 0010

Setting data number (SD no.) and setting data bits, dependent on type of peripheral device

© Siemens AG 1991 All Rights Reserved

Peripheral device	Ident. of the device type (CRT display)	SD no. Interface 1	SD no. Interface 3	Required setting data bits (binary code)
Siemens Printer PT 88 RS232C (V.24), 9600 baud	RTS - LINE	5012 5013	5032 5033	0000 0000 1100 0111
SINUMERIK System 800 NC-NC Coupling RS232C (V.24), 9600 baud	RTS - LINE	5010 5011 5012 5013 5016	5030 5031 5032 5033 5036	0000 0000 1100 0111 0000 0000 1100 0111 0010 0010
Siemens Programming Workstation PD..., PG... RS232C (V.24), 4800 baud	PD / PF	5010c) 5011 5012 5013 5014 5015	5030 5031 5032 5033 5034 5035	0000 0011 1100 0110 0000 0011 1100 0110 0001 0001 1001 0011
Sanyo cassette M2502U-ZE601 RS232C (V.24), 1200 baud	RTS - LINE	5010 5011 5012 5013	5030 5031 5032 5033	0000 0000 1100 0100 0000 0000 1100 0100
Teletype, ASR33 Full duplex 20 mA, 110 baud	RTS - LINE	5010 5011 5012 5013	5030 5031 5032 5033	0000 0000 1100 0000 0000 0000 1100 0000
Facit reader 4040, 4042 Reader/punch RS232C (V.24), 1200 baud	RTS - LINE	5010 5011 5012 5013	5030 5031 5032 5033	0000 0000 1100 0100 0000 0000 1100 0011
Facit reader 4030 RS232C (V.24), 1200 baud	RTS - LINE	5010 5011	5030 5031	0000 0000 1100 0100

Setting data number (SD no.) and setting data bits, dependent on type of peripheral device

c) up to and including Software Version 6

© Siemens AG 1991 All Rights Reserved

Peripheral device	Ident. of the device type (CRT display)	SD no. Interface 1	SD no. Interface 3	Required setting data bits (binary code)
Sommer terminal MDC-3 SNC Cassette unit RS232C (V.24), 9600 Baud	RTS-LINE	5010 5011 5012 5013	5030 5031 5032 5033	0000 0000 1100 0111 0000 0000 1100 0111
VTE 340 Emulation RS232C(V.24), 9600 Baud	RTS-LINE	5010 5011 5012 5013		0001 0010 1100 0111 0001 0010 1100 0111

Setting data number (SD no.) and setting data bits, dependent on type of peripheral device

d) from GA2, Software Version 1

© Siemens AG 1991 All Rights Reserved

6.4 Device connection data

SINUMERIK reader T40 and T50

Cable Order No.: 6FC9 340-8S.

Device data

Transfer rate	9600 baud
Character format	1 start bit 8 data bits 2 stop bits

Settings on T40 reader

Switch block A:

1	2	3	4	5	6	7	8
ON	ON	-	OFF	OFF	OFF	OFF	OFF

Switch block B:

1	2	3	4	5	6	7	8
ON	OFF	OFF	OFF	OFF	OFF	OFF	OFF

Settings on T50 reader

Jumper assigment at 27 PO1:
not allocated

Jumper assigment at 27 PO2:
Jumpers 2 and 5 closed

Jumper assigment at 27 SO2:
Jumpers 1, 2, 3, 4 open

Operating notes

To automatically start from the SINUMERIK, the reader must be ready to start, the "Reader Start" LED must be lit. An exact character stop is not possible.
If programs are produced externally, and several programs punched onto one tape, about 20 blanks must be left between programs.
When punching programs from the SINUMERIK, these blank characters are automatically generated, provided that the "Output without leading and trailing tape" setting data bit is not set
(SD no. 5016, bit 1 for interface 1 or SD no. 5024, bit 1 for interface 2).

SINUMERIK reader T60 (Hand-held unit) Type GNT 2910

Cable Order No.: 6FC9 344-2C.

Device data

Transfer rate	9600 baud
Character format	1 start bit 8 data bits 2 stop bits

Settings

DIP switch in unit

1	2	3	4	5	6	7	8
OFF	OFF	OFF	OFF	OFF	OFF	ON	OFF

Operating notes

Before switching the device on, ensure that the paper tape has been inserted. If there is a fault, the green LED flashes. Acknowledge by switching the device off and on. Read-in start is controlled by the SINUMERIK.

Further operating conditions

The device stops on a precise character.

SINUMERIK WS 800

Cable Order No.: 6FC9 344-1B. (RS232C (V.24))
6FC9 344-1Q. (20 mA)

Device data

Interface RS232C (V.24) or
20 mA current loop (TTY)

Transfer rate	9600 baud
Character format	1 start bit 8 data bits 2 stop bits

Settings on the interface module DF 20

Interface SS2

RS232C (V. 24): no hardware setting

TTY: passive
Socket X9
Jumper 2 - 14 Receiver
3 - 13
6 - 10 Transmitter
7 - 9

© Siemens AG 1991 All Rights Reserved

Siemens Programmer Unit PG 685/675/670

Interface **S5 PLC**

Cable Order No.: 6FC9 340-8G

Device data

Interface:	20 mA current loop
Transfer rate:	9600 baud
Character format:	1 start bit, 8 data bits, 1 parity bit (even parity), 2 stop bits

Operating conditions

For PLC programming the PG 685**/**675**/**670 is connected directly to Interface 3 (20 mA current-loop interface) on the SINUMERIK.

Siemens Programmer Unit PG675

Interface **Printer**

Cable Order No.: 6FC9 344-1A.

Device data

Interface	RS232C (V.24)
Transfer rate	1200 baud
Character format	1 start bit 8 data bits 2 stop bits

Operating conditions

Archiving of NC cyles and programs onto disk, and transfer to and from the NC and the creation of programs is possible with the PG 675. For data transfer, the PG 675 should be connected from the **printer** interface to an RS232C (V.24) interface on the SINUMERIK.

Siemens Programmer Unit PG 615 U with PG 615 adapter and power pack

Interface **PLC**

Cable Order No.: 6FC9 340-8H.

Device data

Interface	20 mA current loop
Transfer rate	9600 baud
Character format	1 start bit 8 data bits 1 parity bit (even parity) 2 stop bits

Operating conditions

For PLC programming, the PG 615 is connected via the adapter and a cable directly to interface **3** (20 mA current-loop) on the SINUMERIK.

© Siemens AG 1991 All Rights Reserved

Siemens page printer PT80

Cable Order No.: 6FC9 340-8C **.** RS232C (V.24)
6FC9 340-8T **.** (20mA)

Device data

Transfer rate	300 baud
Character format	1 start bit 8 data bits 2 stop bits

Order No. for PT80 to SINUMERIK specification:

Type RS232C (V.24):	L22751-A80-D442 (Interface module STT104)
Type 20mA	L22751-A80-D441 (Interface module STT104+LAT101)
	Additional cable for terminal connection: 6FC9340-4KA

An NC-controlled read operation (start/stop) is possible on the device with a 20mA interface.

Siemens printer PT88

Cable Order No.: 6FC9 340-8D.

Device data

Interface connection SAP-S1 (RS232C (V.24))

Setting of the operating mode switches

Switch S1

1	2	3	4	5	6
ON	ON	ON	OFF	OFF	OFF

(with this setting, the printer has a transfer rate of 9600 baud)

Switch S2

1	2	3	4	5	6
OFF	ON	OFF	ON	OFF	ON

(this setting denotes:
BUSY (X2.10) line is switched to BUSY (X1.25) line, with negative potential)

© Siemens AG 1991 All Rights Reserved

7 Appendix

7.1 List of abbreviations

Abbreviations	Meaning
BCD	Binary coded decimals
CH1...CH8 (CH16)	Channel (16 channels for SINUMERIK 880)
CLF	Clear file
COP ON	Switch on coupling
COP OFF	Switch off coupling
CPU	Central processing unit
CRC	Cutter radius compensation
DAC	Digital/analog converter
DIO	Data input/output
DRF	Differential resolver function
EIA Code	Special punched tape code, uneven number of holes per character
ELG	Electronic gear
EOB	End of block, marks the end of the block with data in EIA code
EOR	Marks the program number in EIA code
EPROM	Program memory with program burnt in
EZS	Input buffer memory
FA	Following axis
FS	Following spindle
IKA1	IKA relations (interpolatory compensation with absolute values)
IKA2	Error curve pointer (interpolatory compensation with absolute values)
IKA3	Compensation points (interpolatory compensation with absolute values)
INC	Incremental mode
ISO Code	Special punched tape code, even number of holes per character
LA	Leading axis
LED	Light emitting diode
LF	Marks the end of the block with data in ISO code
LS	Leading spindle

© Siemens AG 1991 All Rights Reserved

Abbreviations	Meaning
MD	Machine data
MDA	Manual data input/automatic
MDI	Manual data input
MPF	Main program file (NC part program)
NC	Numerical control
NC START	Start program
NC STOP	Stop program
PCF	PLC error message texts (PLC files)
PG	Programming unit
PLC	Programmable logic control
Pr. Stop	Display of programmed interruption of the program sequence
RAM	Program memory which can be written into and read from (random access memory)
RPA	R parameter numbers with value assignment (R parameter active)
SD	Setting data
SEA	Addresses with value assignment (setting data active)
SEA4	Setting data active 4
SPF	Sub program file (subroutine)
SW limit switch	Software limit switch
TEA1	NC machine data (testing data active)
TEA2	PLC machine data (testing data active)
TEA3	IAR machine data (testing data active)
TEA4	Cycle setting data (testing data active)
TNRC	Tool nose radius compensation
TO	Tool offset
TOA	Tool compensation (tool offset active)
TOG	Simulation tool data (tool offset active)
UMS	User memory submodule
ZO	Zero offset
ZOA	Zero offset (zero offset active)

© Siemens AG 1991 All Rights Reserved

7.2 List of terms used

Term	Section
Acceleration limitation synchronous	3.2.6/3.2.7
Acknowledge alarm	2.1.1.8
Actual position	2.1.1.8
Address/numeric key	2.1.1.3
Alarm messages	4.6
Angle of rotation	3.2.8
Angle offset	3.2.7
Approach reference point	2.3.2/3.2.2
AUTOM. TO	2.5
AUTOM. ZERO POINT	2.5
AUTOMATIC	3.2
”Axial” setting data	3.2.6
AXIS CONVERTER	2.5
Axis converter	3.2.8
Back-up batteries	5.3
Basic dimension	3.2.4
BLANK	2.5
BLANK AREA	2.5
BLOCK END	2.5
BLOCK NUMBER	2.5
BLOCK SEARCH	2.5
Block search with calculation/automatic block search	3.3.3.4
Channel switchover	2.1.1.8
CIRCLE	2.5
Cleaning	5.5
CLEAR PRE-OFF	2.5
COMPENSATION DATA (COMPEN. DATA)	3.2.4.5
Contour definition	3.2.8
CONTOUR DEFINITION	2.5
Control keys	2.1.1.8

© Siemens AG 1990 All Rights Reserved

Term	Section
CONVERTER	2.5/3.2.8
COPY	2.5
Copy program	3.2.12.4
CORR. BLOCK	2.5
Correction and input keys	2.1.1.7
Correction display	3.2.11.1
CORR. ON-OFF	2.5
CRT display	2.1.1.1
Cursor	2.1.1.8
CURRENT BLOCK	2.5
Current block	3.3.2
CURRENT VALUES	2.5
Current values	3.3.2
Cycle protection	3.2.12.2 3.2.12.3
Copy cycles	3.2.12.5
CYCLES	2.5
DATA IN-OUT	2.5
DATA IN START	2.5
Data input/data output	3.2.12
DATA OUTPUT	2.5
DEC. SBL YES-NO	2.5
DELETE	2.5
Delete input	2.1.1.4
Delete program	3.2.12.8
Delete word/block	2.1.1.7
Device connection data	6.7
Device setting data	6.6
DIAGNOSTICS	2.5
Differential resolver function (DRF)	3.3.9

© Siemens AG 1991 All Rights Reserved

Term	Section
Direction keys	2.1.2
DIRECTORY	2.5
DRF	2.5
DRF YES-NO	2.5
DRY RUN YES-NO	2.5
Dry run feedrate	3.2.8
EDIT	2.5
Editing and input keys	2.1.1.7
Electronic gear	3.2.6
Emergency STOP	2.1.2
Emergency retraction	3.2.6/3.2.7
EXECUTION FROM EXTERNAL	3.3.3.2
External machine control panel	2.1.2
FA overlay	3.2.6
FEED	2.5
Feed stop/feed start	2.1.2
Feed/Jog	2.3.2
Feedrate	2.1.2
Feedrate/Rapid traverse override	2.1.2
Fictitious axis	3.2.6
FILE TRANSFER	2.5
Following axis	3.2.6
Following spindle	3.2.7
FS overlay	3.2.7
Geometry data	3.2.4.1
G FUNCTIONS	2.5
Identifier number	3.2.4
Incremental feed 1... 10 000	2.3.2
"Incremental" (INC...) mode	3.3.5.2
Insert character/word	2.1.1.7
Insert/delete block	3.2.11

© Siemens AG 1990 All Rights Reserved

Term	Section
Installation	2.1.1.8
Interfaces	6
Interrupting the program	3.3.3.4
"Jog" mode	3.3.5.1
Keyswitch	2.1.2
Leading axis	3.2.6
Leading spindle	3.2.7
LED panel	2.1.1.2
LED: ”Feed hold”	2.1.1.2
LED: ”Monitoring active”	2.1.1.2
LED: ”Position not yet reached"	2.1.1.2
LED: "Program running"	2.1.1.2
LINE	2.5
MAIN PRG. START	2.5
Maintenance	5
Manual data input**/**Automatic	2.3.2
MDI**/**Automatic (MDA)	3.3.4
MEASURE TOOL	2.5
Menu tree	2.3.5
Menu trees for operating modes	2.4
MISC. FUNCTIONS	2.5
Modal	2.1.2
Mode selection	2.1.2**/**2.3.3
Modify**/**insert**/**delete word	3.2.11
Modify word	2.1.1.7
Monitoring	4
MOVE	2.5
NC ALARM	2.5
NC MD	2.5
NC STOP**/**NC START	2.1.2

© Siemens AG 1991 All Rights Reserved

Term	Section
Operating data	5.1
Operating mode menu trees	2.4
Operating modes	2.3
Operating modes-Overview	2.3.2
Operating Sequences	3
Operating sequences, machining	3.3
Operating sequences, preparation	3.2
Operator interface	2.1.1
Operator prompting on input	3.2.7.2
Options	2.1
Overstore	3.3.3.2
OVERSTORE	2.5
Paging	2.1.1.8
PART PROGRAM	2.5
Part program start	3.3.1
Playback	3.3.6
PLAYBACK	2.5
PLC ALARM	2.5
PLC MD	2.5
PLC MESSAGE	2.5
PLC STATUS	2.5
PRG. STP. ON/OFF	2.5
PROG. HANDL.	2.5
PROGRAM CONTROL	3.3.3.1
PROGRAM CONTROL MODIFICATION	2.5
Program editing	3.2.9
PROGRAM END	2.5
Program input	3.2.7
Program interruption	3.3.3.5
Program management	3.2.10

© Siemens AG 1990 All Rights Reserved

Term	Section
Program memory	3.2.10.8
PROMPTING	2.5
Prompting on input	3.2.9.2
R Parameter	3.2.6
R PARAMETER	2.5
”R Parameter” setting data	3.2.6
Rapid traverse	2.1.2
Real axes	3.2.6
Rearrange program memory	3.2.12.9
Remote PG	3.2.14.2
RENAME	2.5
Rename program	3.2.12
REORG	2.5
Repositioning	2.3.2
Repositioning (Repos)	3.3.11
Reset	2.1.2
RESET	2.5
”Reset” with change of operating mode	2.3.4
RS232C (V.24) interface	6.1
Scratch	3.3.12
Search for address, block no., word or data	2.1.1.8
Select program	3.2.12.10
Selection of operating modes	2.1.2.1/2.3.3
SERVICE AXES	2.5
Set actual value	2.3.2
Set actual value (Preset)	3.3.10
Setpoint suppression	3.2.6/3.2.7
Settable zero offsets	3.2.5.1
Setting data	3.2.5/3.2.6/6.4
SETTING DATA	2.5
Setting data bits	3.2.14.1

© Siemens AG 1991 All Rights Reserved

Term	Section
Setting data R parameters	3.2.6
Setting data spindle	3.2.6
Setting data zero offset	3.2.5
SIMULATION	2.5
Simulation	3.2.11
Simulation data	3.2.13.3/4
Simulation sequence	3.2.13.5
Simulation tool data	3.2.13.2
Simulation window	3.2.13.6
Single block	2.1.2
SINGLE BLOCK	2.5
SKIP BLOCK YES-NO	2.5
Smoothing constant with thread	3.2.8
Softkey functions	2.5
Softkeys	2.1.1.1
SPECIAL BLOCK	2.5
SPECIAL FUNCT.	2.5
SPINDLE	2.5
SPINDLE CONVERTER	3.2.8
Spindle setting data	3.2.6
Spindle speed	2.1.1.6
Spindle speed limitation	3.2.8
Spindle speed override	2.1.2
Spindle stop	3.2.8
Spindle stop/spindle start	2.1.2
START	2.5
Start angle	3.2.8
STOP	2.5
STORE	2.5
STORE AXIS	2.5
SUBROUTINE	2.5

© Siemens AG 1990 All Rights Reserved

Term	Section
SUBROUTINE START	2.5
Synchronisation	3.2.6
Synchronisation parameter	3.2.6
Synchronisation position	3.2.6
Synchronism	3.2.6/3.2.7
Synchro-spindle	3.2.7
Switching off	2.2.2
Switching on	2.1.2/2.2.1
Teach in	3.3.5
TEACH IN	2.5
THREAD	2.5
Thread recutting	3.2.4.4
TOOL	2.5
Tool number	3.2.4
TOOL OFFSET	2.5
Tool offsets	3.2.4
Tools	3.2.3
Transformation	2.3.4.2
Transmission parameter	3.2.6/3.2.7
Transmission ratio	3.2.6/3.2.7
Universal interface	2.1.2
UPDATED BLOCK	2.5
UPDATED VALUES	2.5
Wear data	3.2.4.1
Zero offset	3.2.5
ZERO OFFSET	2.5

© Siemens AG 1991 All Rights Reserved

NOTES

© Siemens AG 1990 All Rights Reserved

7.3 SINUMERIK 880 Operator interface- – Layout of control and machine control panel

© Siemens AG 1991 All Rights Reserved

Siemens AG

AUT V250
P. O. Box 48 48
W-8500 Nuremberg 1
Federal Republic of Germany

Suggestions

Corrections

For Publication**/**Manual:

SINUMERIK 880, SW 6
SINUMERIK 880 GA2, SW 1

Operator's Guide

User Documentation

Order No.: 6ZB5 410-0HC02-0BA1
Edition: January 1993

From

Name

Company**/**Dept.

Address:

Telephone: **/**

Should you come across any printing errors when reading this publication, please notify us on this sheet. Suggestions for improvement are also welcome.

Suggestions and/or corrections

Siemens AG
Automation Group
Automation Systems
for Machine Tools, Robots
and Special-Purpose Machines
P. O. Box 31 80, W-8520 Erlangen
Federal Republic of Germany

© Siemens AG 1991 All Rights Reserved
Subject to change without prior notice

Siemens Aktiengesellschaft

Order No. 6ZB5 410-0HC02-0BA1
Printed in the Fed. Rep. of Germany
251/332135-14363 BA 05930.4

Progress
in Automation.
Siemens