SINUMERIK 805
Software Version 4

Operator's Guide 05.93 Edition

Supplement

Order No.: 6ZB5 410-0EE02-0BN2

These sheets constitute a **supplement** to the edition

SINUMERIK 805
Software Version 4
Operator's Guide

11.91 Edition Order No.: 6ZB5 410-0EE02-0BA2

The supplement refers to Sections 2,4,5, 6 and 8

Please replace the following pages:

Page	Replace	Insert
Inside title page/ Printing history page	X	
2-5 to 2-8	X	
2-17 and 2-18	X	
2-23 and 2-24	X	
2-27 and 2-28	X	
2-31 and 2-32	X	
2-35 and 2-36	X	
4-1 and 4-2	X	
5-21 and 5-22	X	
5-63 to 5-66	X	
5-71 and 5-72	X	
6-11 to 6-14	X	
8-27 and 8-28	X	

Siemens AG
Automation Group
Automation Systems for Machine Tools, Robots
and Special-Purpose Machines
91050 Erlangen

Subject to change without prior notice

Siemens Aktiengesellschaft

Printed in the Fed. Rep. of Germany

SINUMERIK 805
Software Version 4

Operator's Guide

User Documentation

May 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.

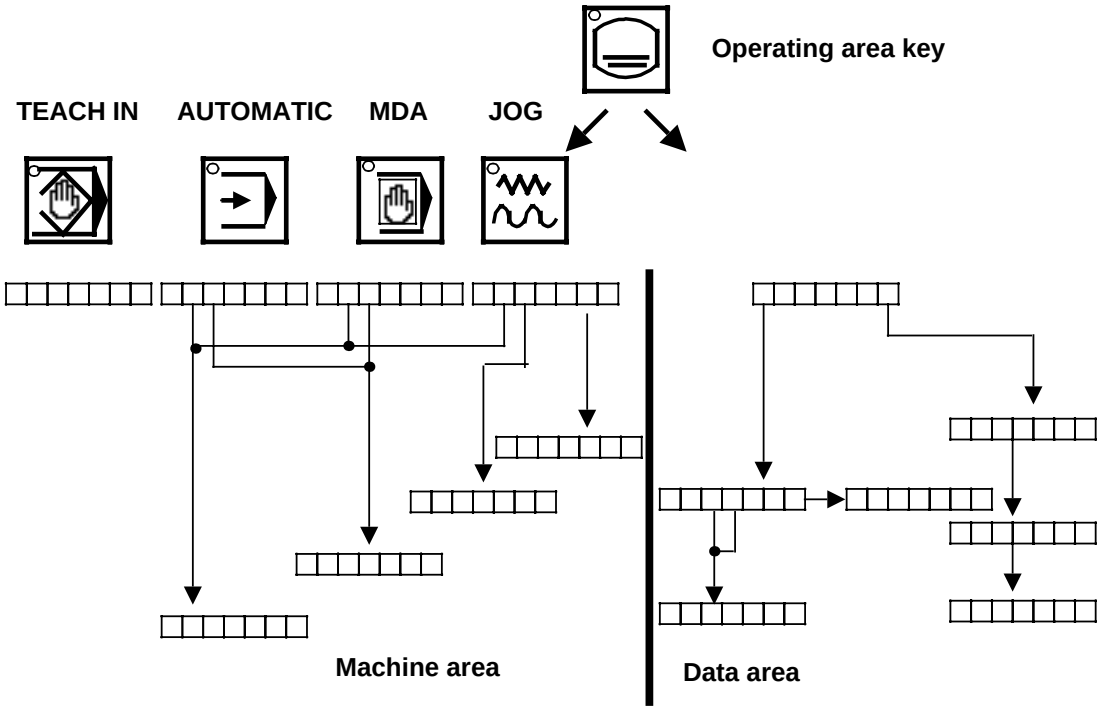
Status code in "Remarks" *column*:

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
06.90	6ZB5 410-0EE02-0BA0	A
01.91	6ZB5 410-0EE02-0BA1	C
11.91	6ZB5 410-0EE02-0BA2	C
05.93	6ZB5 410-0EE02-0BN2	Supplement

Operating area key

This key is used to switch between the two operating areas of the SINUMERIK 805, the machine area and the data area.



- You are in the machine area for mode dependent functions (LED not on):
  Press this key in any machine area menu to screen the basic menu for one data area (depends on NC machine data).
- You are in the data area with functions for displaying and changing data (R parameters etc.) and NC programs (LED on):
  Press this key in any data area menu to screen the basic menu for the selected mode.

JOG key

When you operate this key, you can select the JOG mode from the machine area or the data area. The LED assigned to the key is on for as long as the mode is in force. The selected mode is also displayed in the working field on the screen.

TEACH-IN key

Press this key in the machine or data area to invoke the TEACH IN mode.

The LED assigned to this key stays on as long as you are in TEACH In mode. The mode is also displayed in the on-screen working field.

MDA key

When you operate this key, you can select MDA mode in the machine area and data area. The LED assigned to the key is on for as long as the MDA mode is in force. The selected mode is also displayed in the working field on the screen.

AUTOMATIC key

When you operate this key, you can select AUTOMATIC mode in the machine area and data area. The LED assigned to the key is on for as long as the AUTOMATIC mode is in force. The selected mode is also displayed in the working field on the screen.

SINGLE BLOCK key

This key allows you to execute a part program block by block in the AUTOMATIC mode. Repeated operation of this key switches the function on and off.

- LED out: single-block execution inactive
- LED on: single-block execution active
  - STOP: SINGLE BLOCK appears on the screen
  - The current block is processed when the NC start key is pressed; STOP: SINGLE BLOCK disappears from the screen while the block is being processed.
  - Once the block has been processed, STOP: SINGLE BLOCK is redisplayed.
  - The next block is processed when the NC start key is pressed, and so on.

NC STOP key

By pressing this key, you can interrupt the execution of the current part program in the AUTOMATIC mode. During this interruption, the LED assigned to this key is on and STOP:AUTO-Interruption is displayed on the screen.
Press the NC start key to resume the program at the point of interruption.

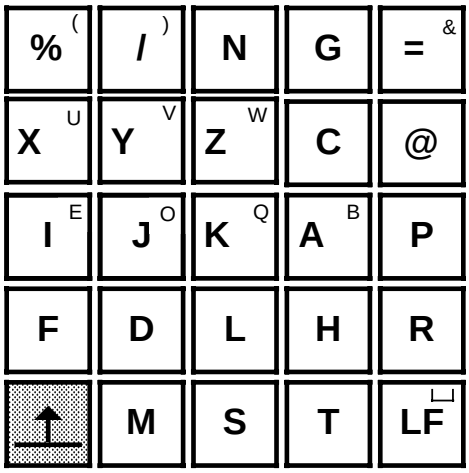
The LED goes out and the display disappears from the screen.

NC START key

- AUTOMATIC mode:
  The selected part program is started at the current block. Whilst the program is being executed, the LED assigned to this key is on and PROGRAM RUN is displayed on the screen.
- MDA mode:
  Press this key to start execution of the specified NC block.

The axis positions programmed in the current block are approached with linear interpolation. Caution: Danger of collision!

Address keys **3**



Case shift

This key is used to activate the **alternate function** of a multi-function key.

– The yellow LED is on:
  The **alternate function** (upper symbol) is active.
– The yellow LED is not on:
  The **primary function** is active.

% (

Symbol: left parenthesis

Character for "start of program"

/)

Symbol: right parenthesis

Sign for "skip block"

N

Address for "block number": N.../letter N

G

Address for "preparatory function": G.../letter G

= &

Ampersand

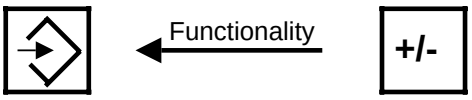
Equal sign

2.1.2 Multi-function keys

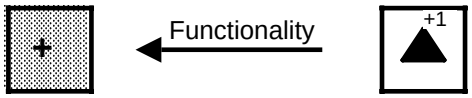
When using software version 4.2 or a newer version, the keyboard can be replaced by a SINUMERIK slimline operator panel.

The operator panel does not have the same layout as the keyboard (some special keys have been omitted).

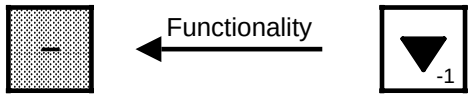
Beginning with software version 4.2, a number of keys therefore have multiple functions:



The INPUT key can perform the same activation/deactivation function (checkmark function) as the +\-key.



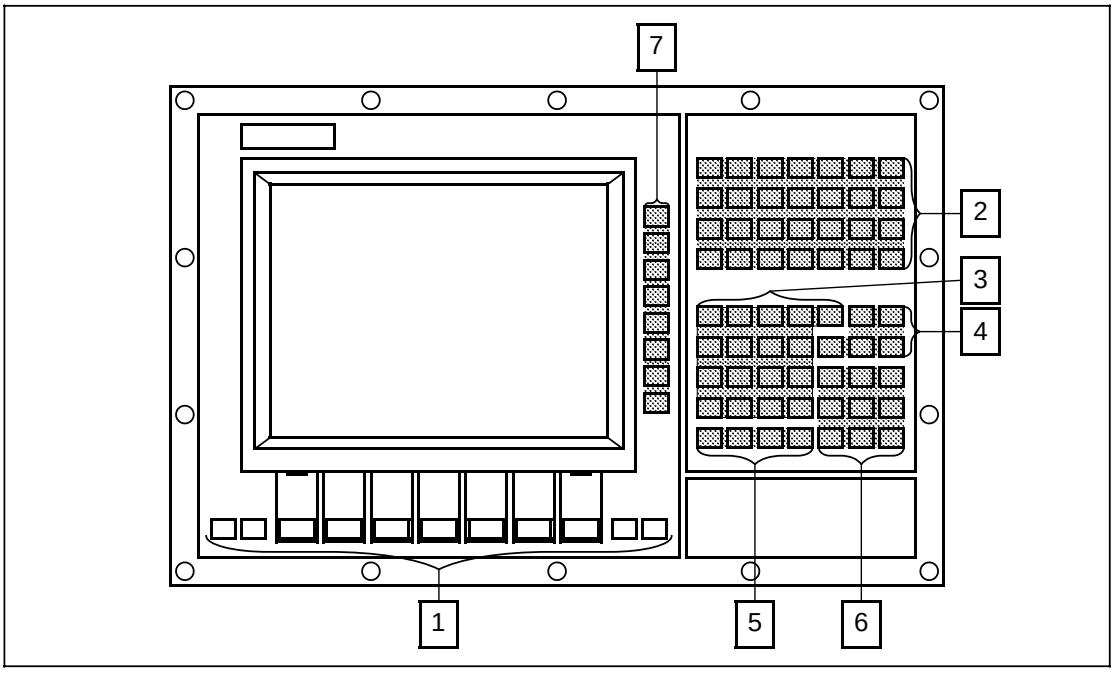
For unopened input fields, the Addition key can perform the same function as the Increment key.



For unopened input fields, the Subtraction key can initiate the same function as the Decrement key.

2.2 Operating elements on the slim-line operator panel

When using software version 4.2 or higher, the keyboard and the monitor can be replaced by a slimline operator panel.

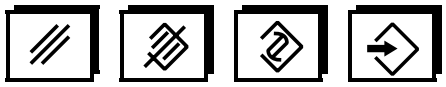


1	Softkeys and menu control keys
2	Alphanumeric keypad
3	Numeric, arithmetic and special symbol keys
4	Control keys
5	Edit and input keys
6	Cursor control keys
7	Special softkeys

Note:

Not all keys on the slimline operator panel call up a function when used with the SINUMERIK 805.

Edit and input keys



Delete character

Using this key to delete characters in an open input field:

- Press the key once to delete the character to which the cursor is positioned.
- Hold down to delete all characters to the right of the original characters.

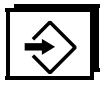
Erase input field

- Use this key to erase the contents of the open input field.
- In program edit mode, use this key to erase words (e.g. M03, X320 or G01).

Open input field

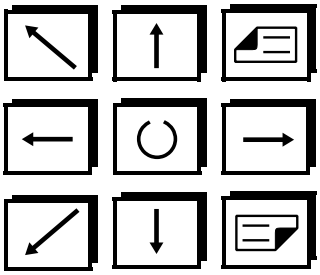
- Use this key to open the input field to which the cursor has been positioned.
  An input field must be opened before an entry can be made.
- When an entry is terminated with this key, the input value is rejected and the original value retained.

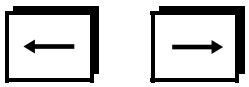
Enter key



- Use this key to confirm your entry:
- In some menus, this key can also be used to activate or deactivate a function selected via the cursor.

Cursor control keys



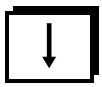


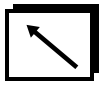
Cursor left, cursor right

- Use these keys to move the cursor left or right from input field to input field.
- Within an open input field, these keys move the cursor from character to character.

Cursor up, cursor down

Use these keys to move the cursor up or down from input field to input field.

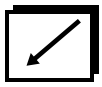




Home key

Use this key to position the cursor

- to the top left input field
- to the start of a part program or data list.



End key

Function not implemented in SINUMERIK 805

Page up, page down

Page down.

Page up.

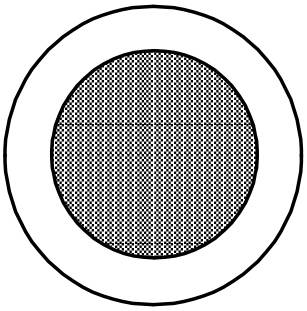
These keys are used to page through multi-page displays.

Release key

Function not implemented in SINUMERIK 805

2.3.1 Description of operating elements on the hand-held unit

Emergency stop button **1**



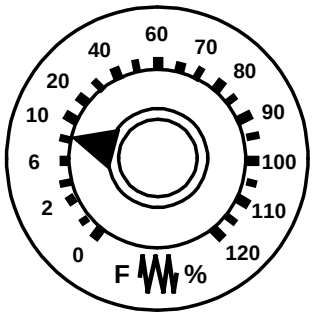
The red button is operated in **emergency situations**:

- when there is a danger to life
- when there is danger of damage to the machine or workpiece

Operation of the "emergency stop" button generally brings all drives to a controlled standstill with maximum breaking power.

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

Feedrate/ rapid traverse override switch **2**



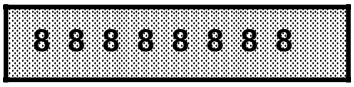
The rotary switch with 23 latched positions enables you to decrease or increase the ***programmed*** feedrate value "F "(corresponds to 100%).

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

Control range:	0% . . . . . 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%

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

Actual axis position and distance to go display **3**



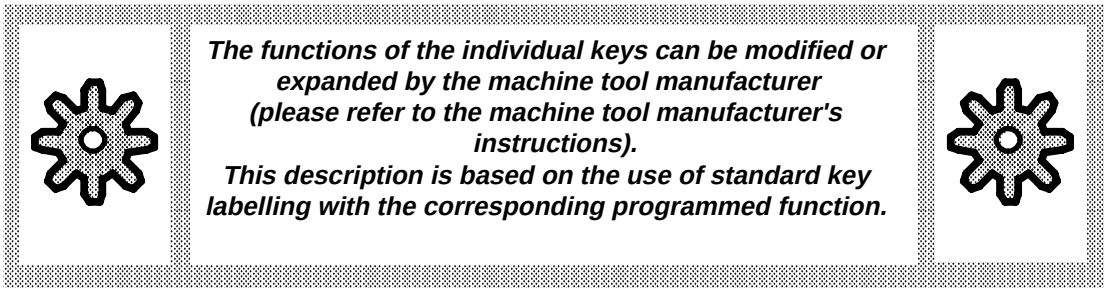
Depending on the selected axis and function, the display shows either the actual position or the distance to go.

Notes:

- The display remains dark if no axis was selected via the PLC/NC interface.
- With software version 4.2 and higher this display can also be used to show PLC data.

Keys with LED display **4**

The key group consists of 25 keys, 16 of which are equipped with LEDs.



The functions of the individual keys can be modified or expanded by the machine tool manufacturer (please refer to the machine tool manufacturer's instructions).
This description is based on the use of standard key labelling with the corresponding programmed function.

Jog key

Press this key to select the JOG mode

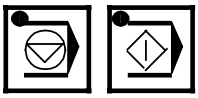
Automatic key

Press this key to select the AUTOMATIC mode.

When the JOG or AUTOMATIC mode is selected, the corresponding key LED goes on. The selected mode is also displayed in the mode field on the screen.

The modes can also be selected by pressing the appropriate keys on the keyboard.

NC stop / NC start



You operate the NC-STOP key. 

- The part program being processed is stopped.
- "STOP:AUTO-INTERRUPTION" is shown on the display.
- The LED in the NC START key goes out.
- The LED in the NC STOP key goes on.

You operate the NC START key in AUTOMATIC mode. 

- The part program called is started and/or continued at the active block.
- "PROGRAM RUNNING" is displayed.
- The LED in the NC STOP key goes out.
- The LED in the NC START key goes on.

The axis positions programmed in the current block are approached with linear interpolation. Caution: Danger of collision!

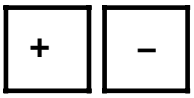
NC STOP and NC START commands can also be issued over the keyboard.

Axis selection keys

The required axis is selected with these keys:

- The actual position of the selected axis is shown in the actual axis position display.
- One of the four key LEDs lights up, showing the selected axis.
- The control defaults to the X axis on power-up.

Axis direction key



Operation of the AXIS DIRECTION PLUS KEY or AXIS DIRECTION MINUS KEY results in the following:

- JOG mode selected:
  The axis traverses in the specified direction.
- Submode JOG INC selected:
  Each time one of these keys is pressed, the selected axis traverses by the set increment in the specified direction.

Rapid traverse override key

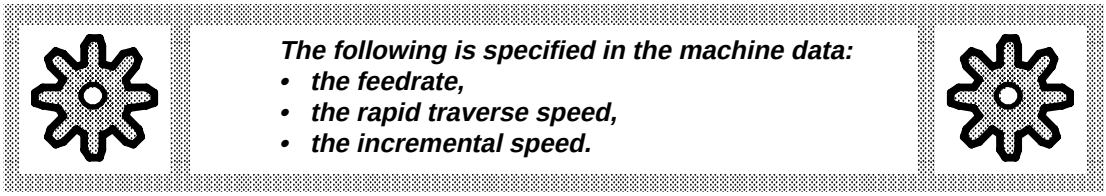
If you press the RAPID TRAVERSE OVERRIDE key

together with the 

or 

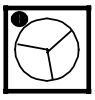
key,

the selected axis travels at "rapid traverse" speed in the specified direction.



The following is specified in the machine data:
- ***the feedrate,***
- ***the rapid traverse speed,***
- ***the incremental speed.***

Handwheel function key



The function is activated/deactivated each time the HANDWHEEL key is pressed. When the handwheel is selected, the appropriate key LED lights up. The HANDWHEEL function affects the selected axis.

2 Display field for program modification (Section 5.3.6)

- STOP: AUTO interruption
- STOP: Single block mode
- STOP: Program STOP M00/M01
- STOP: Read-in enable
- STOP: Dwell time

3 Control display (light background ≙ active) for:

SKP	Skip block
DRY	Dry run
ROV	Rapid traverse override
DBL	Decode single block
DRF	Offset
M01	Programmed stop

4 Status display for FEEDRATE STOP (FST: FEED STOP)
(light background ≙ active)

5 Program number of selected part program

6 Display field for operating status

- RESET
- Program running
- Program interrupted
- NC READY

7 Display field for the number of the selected NC
(for details see Interface Part 1, Signals Flag Byte 99)

8 Area for NC displays: texts, graphics

9 Text bar with 7 softkey functions

10 Message line for NC alarms and PLC interrupts and messages (only the alarm or interrupt with highest priority is displayed ≙ alarm with the lowest number).

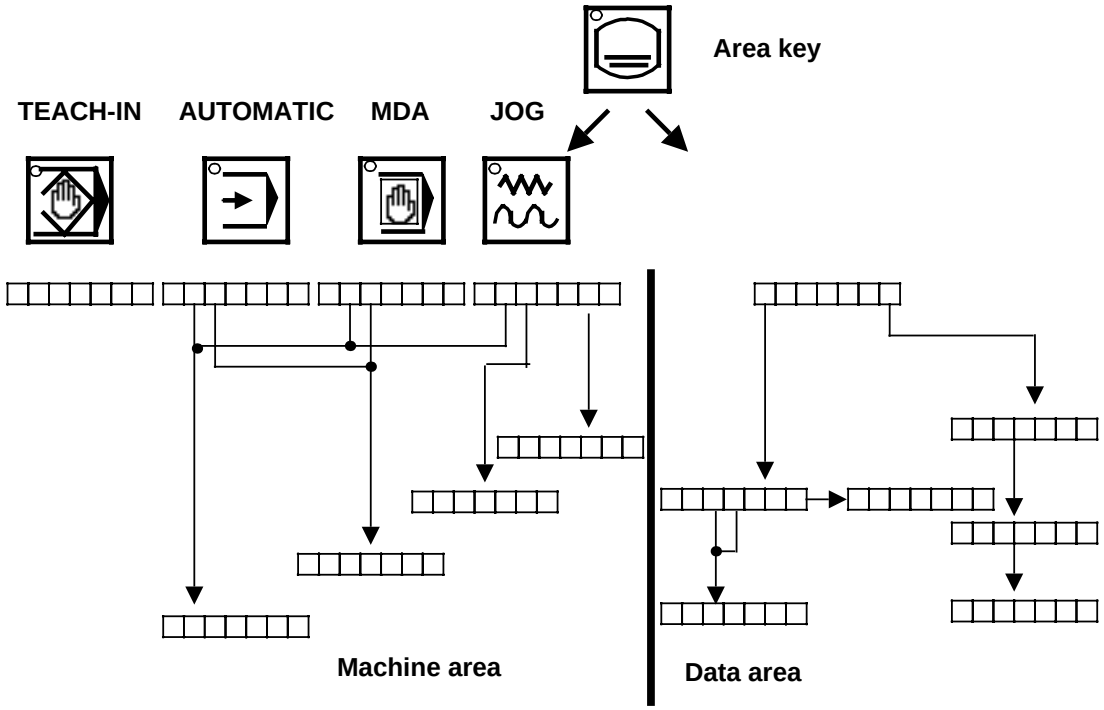
11 Date and time display (Section 4.8.5)

2.5 Areas, menu tree

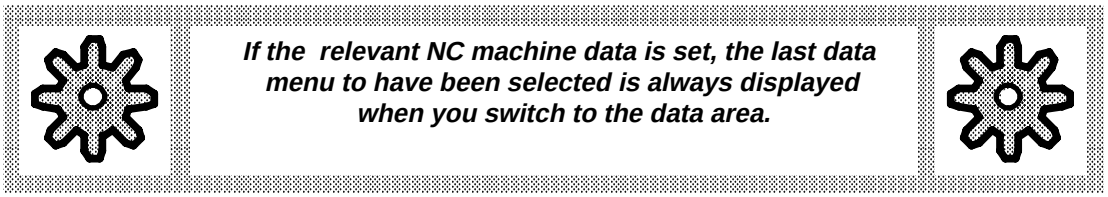
Operation of the SINUMERIK 805 is divided into two areas: The machine area with mode-dependent functions, and the data area, with functions for data and NC program display and modification.

Operation servicing within each area is menu controlled, i.e. the functions shown in the text bar can be selected via function keys (softkeys). As soon as a function has been selected, the control reacts by displaying new functions in the text bar.

The result is a series of multi-branch menu trees.



Using the area key, it is possible to branch to the data area's **basic menu** from any menu in the machine area.



Conversely, it is possible to branch from any data area menu to the basic menu of the selected operating mode.

Comments:

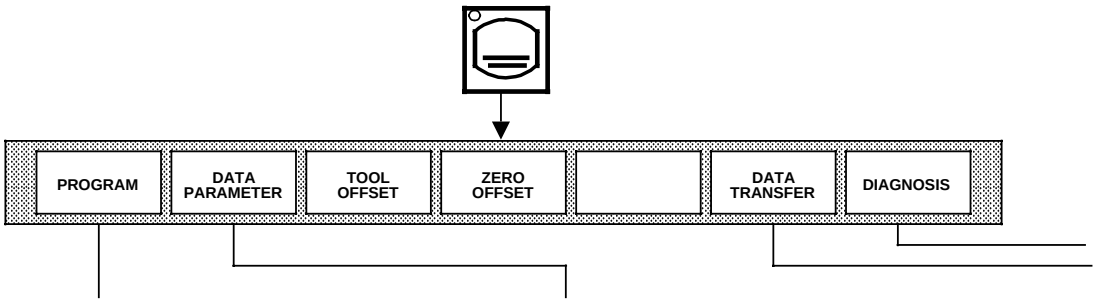
- The key LED lights up to show that the data area has been selected.
- Contrary to the other functions, softkey functions are displayed in reverse video in the text bar.

4 Data Input

4.1 General remarks

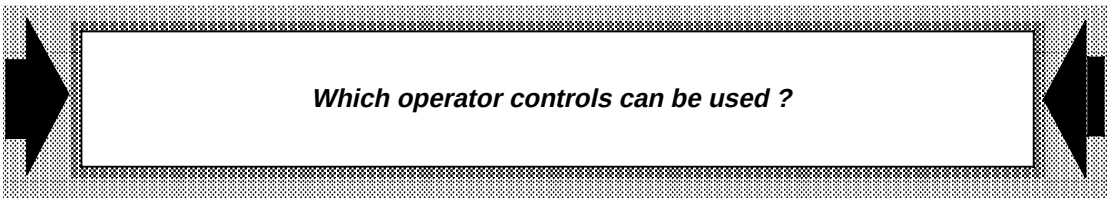
The functions of the various operator controls (Secction 2) and the different modes (Section 3) were discussed in detail in the preceding sections.

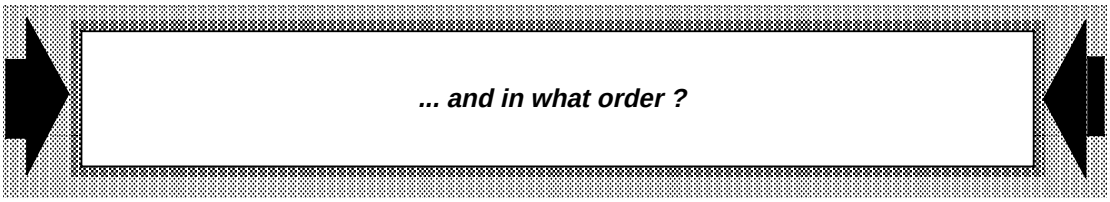
Section 4 explains the input of data for programs, parameters, tool offsets, zero offsets, data transfers and diagnostics.
Section 4 has been structured to conform with the "data area" menu tree:



Section 4 and 5 discuss SINUMERIK 805 operation.

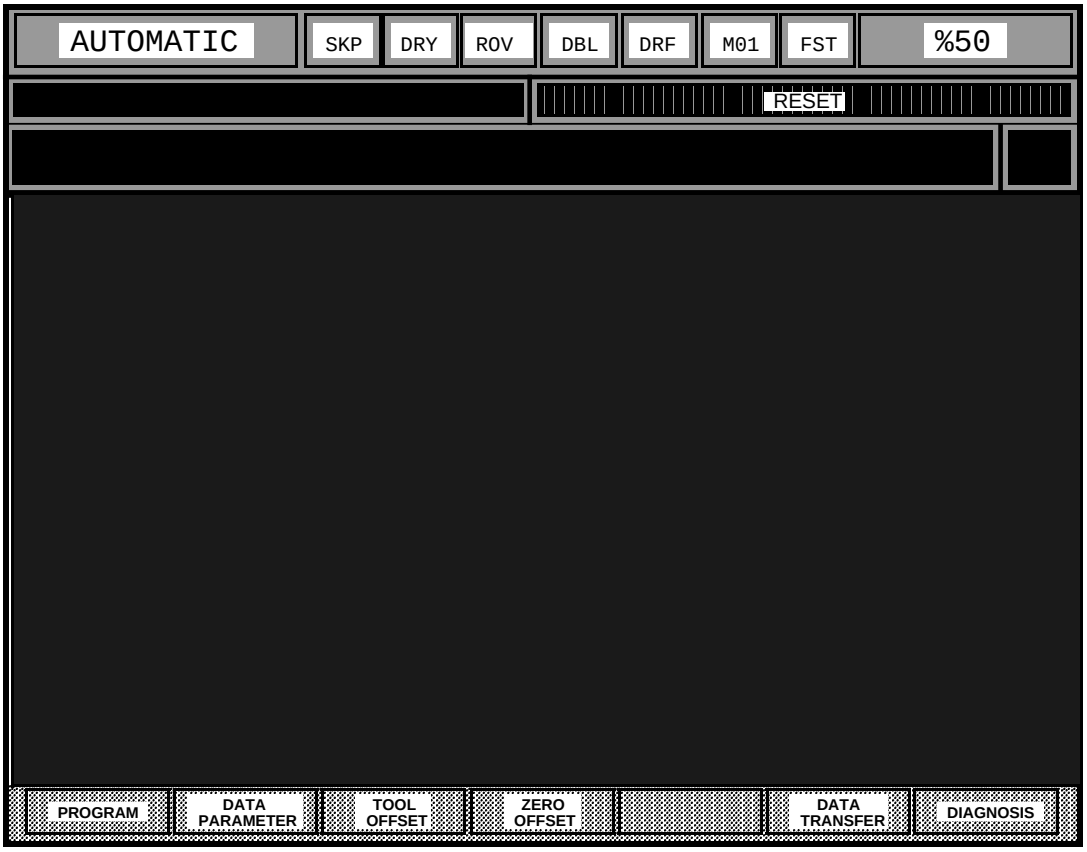
They explain:





4.2 Basic data area display

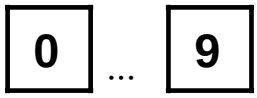
The following is displayed when the SINUMERIK 805 data area is selected via the operating area key.



Notes:

- The LED in the operating area key lights up when the data area has been selected.
- The menu tree is shown in Section 2.5.2.
- See also Section 2.5 (Areas, menu tree).

If the function "user display" is active, a user display comes up on the basic display of the data area, the contents of which the user can configure himself (see NC Programming Guide, Section 12).



Now enter the desired program number with the numeric keys.

Use the INPUT key to close the input field and the window. The selected program (e.g % 50) is displayed in the mode field.

Press the FEED START

and

SPINDLE START keys.

Start the program with the NC start key.

The axis positions programmed in the current block are approached with linear interpolation. Caution: Danger of collision!

Notes:

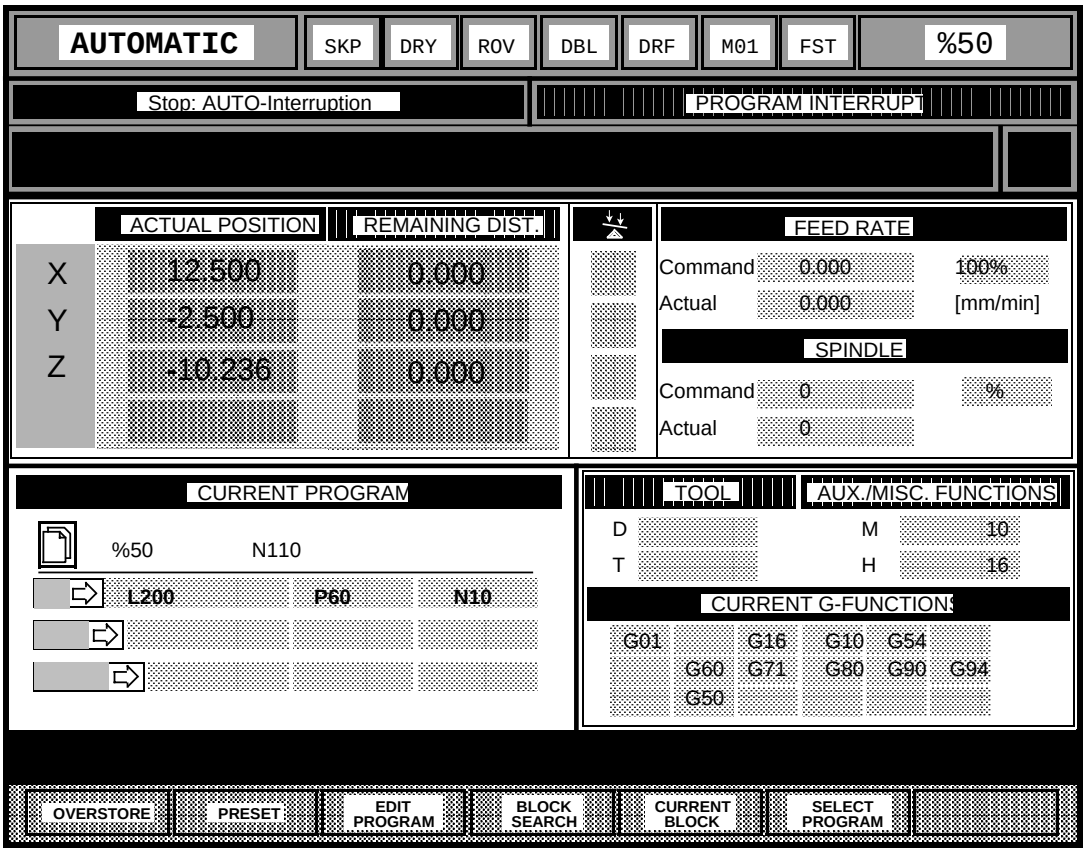
- NC start is not possible if FEED STOP is active.
- You can call up CURRENT PROGRAM or CURRENT BLOCK on the screen while the program is running (see Section 5.3.2).
- Programs can also be selected from the main program and subroutine directories (see Section 4.3.6).

5.3.2 Display of CURRENT PROGRAM or CURRENT BLOCK in AUTOMATIC mode

You can switch between the CURRENT PROGRAM and CURRENT BLOCK display with a softkey.

Operator input sequence

The CURRENT PROGRAM display has been selected. You switch to the CURRENT BLOCK display by pressing the softkey.



5.3.17.2 Creating a new part program

You can choose between two possibilities for creating a program:

- **Manual positioning**

  Use the direction keys or the handwheel to move the axes to the required position. Theaxis position value is forwarded to the program block automatically. Enter all additional functions (preparatory functions, auxiliary functions and the like) over the keyboard.

  Press the EXECUTE BLOCK softkey and the NC START key to output the traversing movement as program block.

  You should program linear or circular movements only. Please use only the commands listed below:

N	Block number
G0, G1	Linear movement
G2, G3, U	Circular movement with given radius
G4	Dwell
G9	Exact stop
G16 to G19	Plane selection
G25, G26	Working area limitation
G53 to G57, G58, G59	Zero offset **Caution!** If the zero offset is modified, the axis is moved accordingly when you press NC START. If G53 to G59 are used, the workpiece related actual value system (setting data) must be active.
G60	Exact stop
G62, G64	Continuous-path mode **Caution!** With G62 and G64, the end point behaviour is different when the part program is processed in Automatic mode.
G68, G90	Absolute dimensioning
G91	Incremental dimensioning
G70, G71	Inch, metric **Caution!** If a G70/G71 change is programmed, conversion inaccuracies may produce slight axis movements.
G92 to G97	F, S definition
F, S, T, M, H, R	

- **Entering the coordinates**

  Enter the coordinates for the target positions. Enter any additional functions (such as preparatory functions, auxiliary functions and the like) over the keyboard. When you press the EXECUTE softkey and NC START, the program block is output and the axes are traversed accordingly.

  Note:

  All G functions are allowed.

  Should you subsequently wish to approach positions using the handwheel and to enter them, any offsets/overrides should first be deactivated and any programmed contour definitions terminated.

Operator input sequence

Prerequisite: You have specified a new main program or subprogram as described in Section 5.3.17.1.

You now have two options:

X Y Z + −

You can traverse the axes to the relevant end point with the axis keys of the handwheel. When an axis comes to a standstill, the axis address and coordinate is displayed on the corresponding input line. A zero offset, if any, is taken into account when the coordinate is computed, as is in active tool length compensation value.

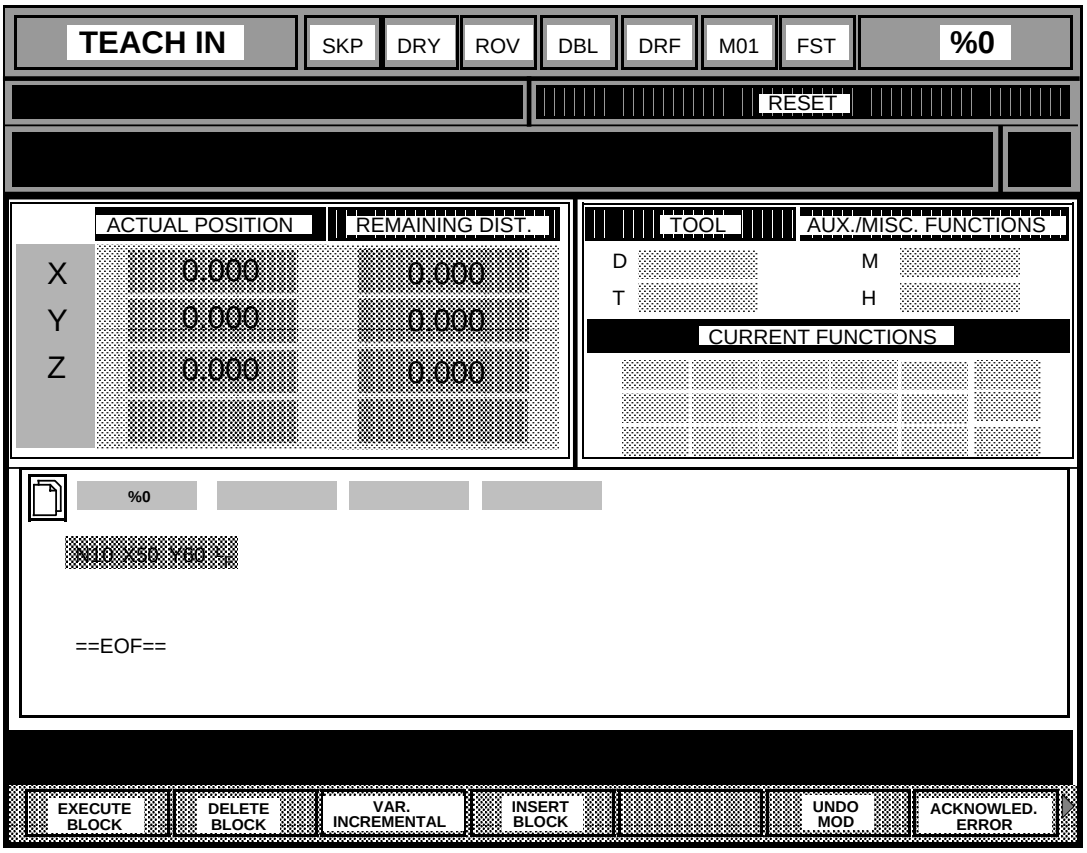
or

A ... Z

0 ... 9

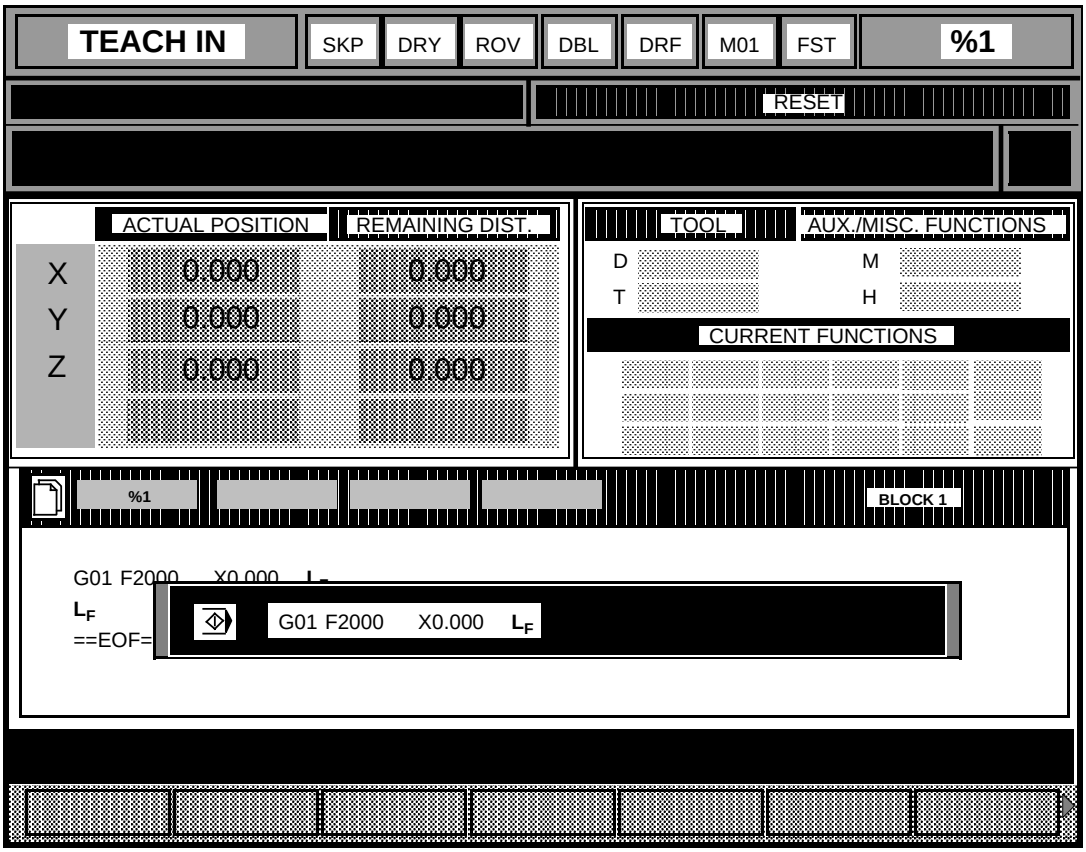
Enter the coordinates for the target positions and the relevant traversing command through the keyboard.

Additional functions (such as preparatory functions, auxiliary functions and the like) are always entered through the keyboard.
Caution: Do not yet press the INPUT key to store the block.



EXECUTE

Press the EXECUTE softkey to confirm your entries for this program block. The block is displayed in a separate window.



The program block just edited is now checked, and the EXECUTE attempt rejected if errors are found.

Press NC START to execute the block.

When the block has executed and no further editing is required, store it by pressing the INPUT key. Once you press INPUT, the block is no longer displayed in reverse video.

The next input line is then opened for entry of the next block.

In TEACH IN mode, editing is possible only on a block-by-block basis, and only in ascending order. The EXECUTE, NC START and INPUT keys must be pressed in this sequence for each block (whether it was edited or not)

Note:

If you want to teach in the axis positions with a different active zero offset, the workpiece related actual value system (setting data) must be active. Please also note the following:

Where axis positions have no active zero offset, G53 must be entered and activated by pressing the softkeys EXECUTE and NC START. Do not store this block as otherwise G53 will no longer be active. Now the axis positions can be "taught in" by traversing the axes. Operate the keys in the sequence EXECUTE, NC START and INPUT to store the block.

If axis positions with a different active zero offset are to be "taught in" in the block immediately following, then the zero offset (also the initial setting zero offset) must first be entered and activated (softkeys EXECUTE, NC START). Only then can the teach-in process for the axes be carried out. The same results can be achieved (activating the last active zero offset) by entering an intermediate block with auxiliary functions and/or traversing distances.

5.3.17.3 Correcting an existing part program

Program blocks can be corrected (modified), inserted and deleted. Modifications, however, are possible only in the highest (first) program level (not in nested subroutines).

In TEACH IN mode, programs on other levels are skipped until a first-level program is located.

a) Correcting from the beginning of the program

Operator input sequence

Prerequisites: An existing main program or subroutine has been selected as described in Section 5.3.17.1.

The following is displayed on the screen:

5.3.17.4 Switch to the data area, another mode or resume processing after an alarm

While in TEACH IN mode, you can switch to the data area or select another mode.

Prerequisite is that the control is at "Stop" and the last block has been stored.

When you select another mode, you exit the TEACH IN mode.

If an error is detected in a block or an alarm or interrupt is generated, press UNDO MOD. to screen the original block.

Operator input sequence

Press RECALL to clear the message. Press the softkey QUIT if an alarm is active.

UNDO MOD.

Press the UNDO MOD. softkey to rescreen the original block (i.e. to view it as it was before you made your changes).

You can now switch to the data area or select another mode.

If you wish to continue processing the program, you must carry out the following operations:

EXECUTE

Press the softkey EXECUTE. The original block is checked.

When NC start is operated the block is executed and the NC returns to the "NC ready"state.

Now you can continue processing the program or block.

"Block not transferred to program memory" is displayed and an internal RESET executed if you exit a subroutine opened in TEACH IN mode when switching to the data area or another operating mode.

If an error was flagged, you should not exit the TEACH IN mode until it has been corrected. To acknowledge the error, press RECALL or RESET.

Note:

When you exit the TEACH IN mode, error messages are displayed when

- the actual position differs from the position last stored
- an entry/correction was made but not stored
- the next block for calculation of an intersection is missing.

5.3.17.5 Error messages

Editing allowed on 1st program level only	
Cause:	The highest (first) program level is not active. G80 is not active.
Effect:	Input is inhibited.
Remedy:	Select highest (1st) program level. Enter G80.

Block not transferred to program memory	
Cause:	The block last entered/modified was not transferred to program memory.
Effect:	The block is not stored whenTEACH IN mode is exited or the next block edited.
Remedy:	Press RESET

Press EXECUTE BLOCK	
Cause:	Entry was not confirmed with EXECUTE softkey.
Effect:	Block is not executed when NC START is pressed.
Remedy:	Press the EXECUTE softkey.

Block not executed	
Cause:	The block was not started or is still executing.
Effect:	The block is not checked for syntax errors or is not processed.
Remedy:	Press NC START or press the EXECUTE softkey after the block has finished executing.

Editing allowed only in TEACH IN block	
Cause:	Characters were entered although the active block was exited with the cursor control keys.
Effect:	Input is inhibited.
Remedy:	Press the TEACH IN softkey or set the cursor to the right block using the cursor control keys.

Set the following parameters using softkeys:

TERMINAL TYPE	Terminal type:	PLC-PROG
BAUD RATE	Baud rate:	9600
STOP BITS	Stop bits:	2
Parity	Parity:	none

After entering the interface parameters, press RETURN to return to the basic DATA TRANSFER menu.

READ-IN START

Press the READ-IN START key. The message "data transfer active" appears on the screen.

The interface has now been activated for PLC communication.

6.5 Interface parameters for I/O devices

Before the SINUMERIK 805 can interchange data with an I/O device, the relevant SINUMERIK 805 must be adapted to that device. This is done by altering the interface parameters in the PARAMETER menu as described in Section 4.7. The table below shows the interface parameters to be set for each of the I/O devices:

Per device / Parameters	Siemens PG 750/685/675 programmer 20 mA SIMATIC S5 Interface	Siemens PG 750/685/675 programmer RS 232 C Printer interface	Siemens PG 615 U programmer 20 mA	Siemens PT 80 page printer RS 232 C/20 mA	Siemens PT 88 page printer RS 232 C
TERMINAL TYPE	PLC-PROG	RTS-LINE	PLC-PROG.	RTS-LINE	RTS-LINE
BAUD RATE	9600	9600	9600	300	9600
STOP BITS	2	2	2	2	2
PARITY	NONE	NONE	NONE	NONE	NONE
X_{ON}					
X_{OFF}					
EIA-Code for @					
EIA-Code for:					
EIA-Code for =					
EIA-Code for end of text		03		03	03
Start with X_{ON}					
Program start using LF					
End of block using CRLF		X		X	X
Data output in EIA code					
Stop using end of text		X		X	X
Evaluate DSR signal					
Leader and trailer		X		X	X
Program from system 3/8					
REORG via interface		X		X	X
Time monitoring		X		X	X

X = Parameter is active

Parameters / Per device	Siemens programming workstation PD.., PG RS 232 C	SINUMERIK System 800 NC-NC link RS 232 C	SINUMERIK reader, T 40, T 50, T 60/RS 232 C	SINUMERIK WS 800 RS 232 C	Teletype ASR 33 full duplex	Sanyo cassette M2502V-ZE-601 RS 232 C
TERMINAL TYPE	PD/PF	RTS-LINE	RTS-LINE	RTS-LINE	RTS-LINE	RTS-LINE
BAUD RATE	4800	9600	9600	9600	110	1200
STOP BITS	2	2	2	2	2	1
PARITY	NONE	NONE	NONE	NONE	NONE	NONE
X_{ON}	11					
X_{OFF}	93					
EIA-Code for @						
EIA-Code for:						
EIA-Code for =						
EIA-Code for end of text						
Start with X_{ON}						
Program start using LF						
End of block using CRLF		X				
Data output in EIA code						
Stop using end of text						
Evaluate DSR signal						
Leader and trailer						
Program from system 3/8						
REORG via interface						
Time monitoring						

X = Parameter is active

Per device / Parameters	Facit 4040, 4042 Reader/punch RS 232 C	Facit 4030 Reader RS 232 C	Sommer terminal MDE-3 SNC Cassette unit RS 232 C
TERMINAL TYPE	RTS-LINE	RTS-LINE	RTS-LINE
BAUD RATE	1200	1200	9600
STOP BITS	2	2	2
PARITY	NONE	NONE	NONE
X_{ON}			
X_{OFF}			
EIA-Code for @			
EIA-Code for:			
EIA-Code for =			
EIA-Code for end of text			
Start with X_{ON}			
Program start using LF			
End of block using CRLF			
Data output in EIA code			
Stop using end of text			
Evaluate DSR signal			
Leader and trailer			
Program from system 3/8			
REORG using interface			
Time monitoring			

X = Parameter is active

3006	**Wrong block structure**	ACKNOWLED.
Explanation:	– More than 3 M functions programmed in the block – More than 1 S function programmed in the block – More than 1 T function programmed in the block – More than 1 H function programmed in the block – More than 4 auxiliary functions programmed in the block – More than 3 axes with G00 / G01 programmed in the block – More than 2 axes with G02 / G03 programmed in the block – G04 has been programmed with addresses other than "X" or "F" – M19 has been programmed with addresses other than "S" – Incorrect or no interpolation parameters for G02/G03 (MD 304*) – The special auxiliary functions for analog output have been incorrectly programmed.	
Remedy:	As for alarm 3000	

3007	**Error in programming setting data**	ACKNOWLED.
Explanation:	– G25/G26 has been programmed – G92 has been programmed with an address other than "S" – M19 has been programmed with an address other than "S"	
Remedy:	As for alarm 3000	

3008	**Subroutine error (M19 missing...)**	ACKNOWLED.
Explanation:	– Subroutine call without number of passes "P" – M30 used as end-of-program – M19 at program end is missing – The fourth nesting depth has been called (SINUMERIK 805 permits only 3 subroutine levels) – M19 has been programmed in main program	
Remedy:	As for alarm 3000	

3009	**Error in part prgram/ part program type**	ACKNOWLED.
Explanation:	Not applicable to SINUMERIK 805	

3010	Intersection error	ACKNOWLED.
Explanation:	This error can occur in connection with stock removal cycle L96 when: – The contour program is written without G0, G1, G2, G3 – @ 714 is used in contour program – Incorrect plane in contour program – No intersection found – More than a quadrant programmed in contour program – No geometry in 1st block of contour subroutine for L95 stock removal cycle	
Remedy:	As for alarm 3000	

3011	Too many axes progr./ axes programmed twice	ACKNOWLED.
Explanation:	– An axis has been programmed twice in the same block – More axes have been programmed than exist in the machine.	
Remedy:	As for alarm 3000	

3012	Block not in memory	ACKNOWLED.
Explanation:	– The program has not been terminated with M02/M03/M17 – The block number stated in the jump (@ 100, 11x, 12x, 13x) has not been found in the specified direction – Non-existent block number specified for block search.	
Remedy:	As for alarm 3000	

3015	Rotary axis not allowed	ACKNOWLED.
Explanation:	– A rotary axis was programmed as simultaneous axis, although this is not allowed. – A standard motor axis was defined as rotary axis.	
Remedy:	– Change the NC program so that the rotary axis is not addressed as simultaneous axis. – Deselect the rotary axis definition	