SINUMERIK 805
Software Version 3

Operator's Guide

User Documentation

January 1991 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**

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 1990 All Rights Reserved

Preliminary Remarks

Reading comments

This document has been written for machine tool users.

The user documentation for SINUMERIK 805 is divided into two parts:

- Operator's Guide
- Programming Guide

The Operator's Guide is divided into three parts:

- **BASICS**: Section 1-3
- **APPLICATIONS**: Section 4-5
- **REFERENCES**: Section 6-9

If this is your first contact with SINUMERIK 805, please start with **BASICS**:

Section 1: **General Comments**
describes

- the SINUMERIK 805 components,
- how to switch SINUMERIC 805 on and off.

Section 2: **Description of Operator Controls and Areas**
explains

- the operator controls and their functions,
- the elements and hierarchical structures contained in the menus in the machine and data areas.

Section 3: **Operating Mode Description**
describes

- how to select the operating modes,
- the components of the operating modes.

The **APPLICATIONS** section contains information about data input, control preparation and operation during the processing phase.

Section 4: **Data Input**
describes

- data input for programs, parameters, tool offsets, zero shifts, data transfer and diagnostics.

Section 5: **Operation Sequences**
explains

- how to operate the sequence before, during and after program input,
- how to operate the sequence during the processing phase.

The **REFERENCES** section gives information about the following:

Section 6: **Data Interfaces**
Section 7: **Maintenance**
Section 8: **Monitoring**
Section 9: **Appendix**

Section 9 contains a softkey function dictionary, and lists of abbreviations and definitions.

Technical comments

This Operator's Guide is valid for Software Version 3!

Contents

APPLICATIONS

REFERENCES

1 General Comments

This Operator's Guide explains the configuration of the SINUMERIK 805 as well as its operation.

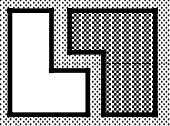
You will be introduced to the following components:

- Screen
- Operator keyboard
- Hand-held unit

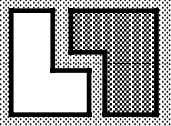
This guide has been devised for the maximum scope of functions offered by the SINUMERIK 805.

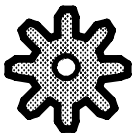
Any required options are to be found in the catalog: NC 34/Options.

Technical comments



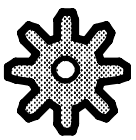
In parts of this documentation you will see the symbol shown on the left and a comment referring to an ordering data option. This symbol is intended to point out that the described function is executable only if the control includes the designated option.





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).

Please consult the machine tool manufacturer's documentation.



- Other functions which are not described in this documentation can possibly also be performed in the control.
- However, the customer is not entitled to demand these functions when new equipment is supplied or when servicing is carried out.

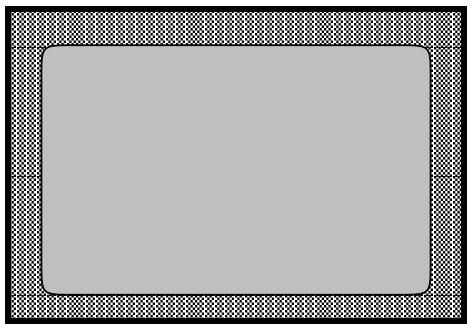
This Operator's Guide is valid for:

SINUMERIK 805, Software version 3.

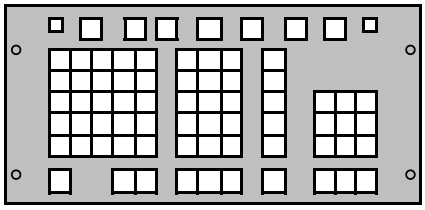
©Siemens AG 1990 All Rights Reserved

1.1 Product

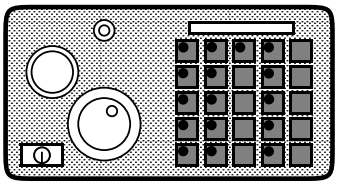
The SINUMERIK 805 is a microprocessor-controlled CNC continuous-path control system for machine tools and automation tasks with the following operator control and display elements:



12" monochrome monitor



Operator keyboard



Hand-held unit

- **Screen displays** provide information on:
  - active operating modes and program modification
  - setpoint/actual values
  - NC and PLC alarms etc.
- The **operator keyboard** is a full keyboard offering (with) the following features:
  - 7 softkeys for selecting various menus in 2 operating areas
  - Key selection of operating modes
  - Cursor keys, numeric keys and double assignment (SHIFT) keys.
- Using the **hand-held unit**
  - JOG and AUTOMATIC modes can be selected
  - the axes can be traversed in jog mode
  - the axes can be traversed using the handwheel.
- In addition:
  - the actual position of the selected axis and
  - the active key function are displayed by means of LED.

The described hand-held unit functions apply to the use of standard labelling strips and a relevant PLC program (see Interface Part 1, Section 4.1 Standard PLC program).

©Siemens AG 1990 All Rights Reserved

1.2 Switching the control on and off

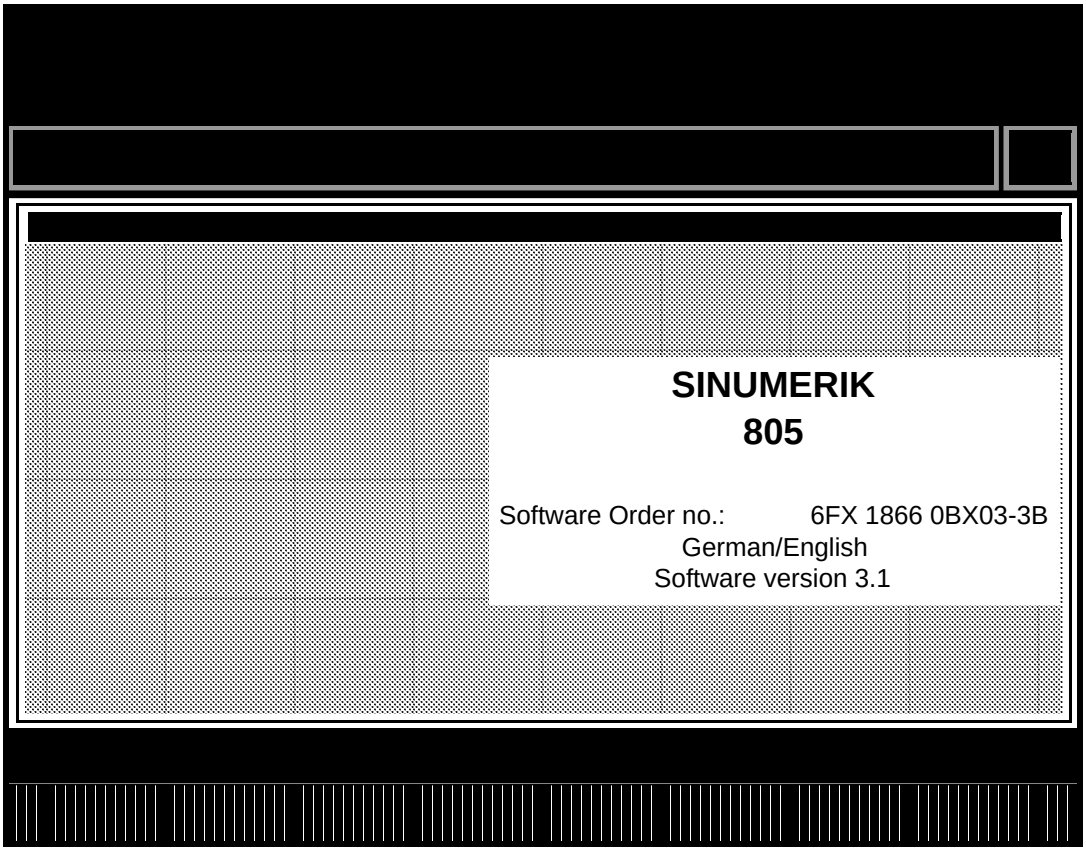
Switching on SINUMERIK 805

There are various ways of switching the control or whole system on, therefore:

Please follow the advice
of the machine tool manufacturer!

In principle it can be said that: the control is switched on when the input voltage applies to the power supply unit of the central controller.

When the control has been switched on, the following system screen appears for approx. 5 seconds, followed by the JOG mode basic display.



Start display after switching on control

©Siemens AG 1990 All Rights Reserved

Comments on system screen:

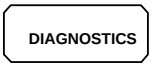
The system screen gives information to the operator regarding the installed software.

6FX 1866 0BX03-3B	:	Order number of the installed system software
Language: German/English	:	Plaintext comments on installed language software
Software version: 3.1	:	Plaintext comments on installed software version

Note:

The system screen can also be called up during operation using the **following sequence of keys**:

Press the operating area key



Press in the following sequence: DIAGNOSTICS softkey

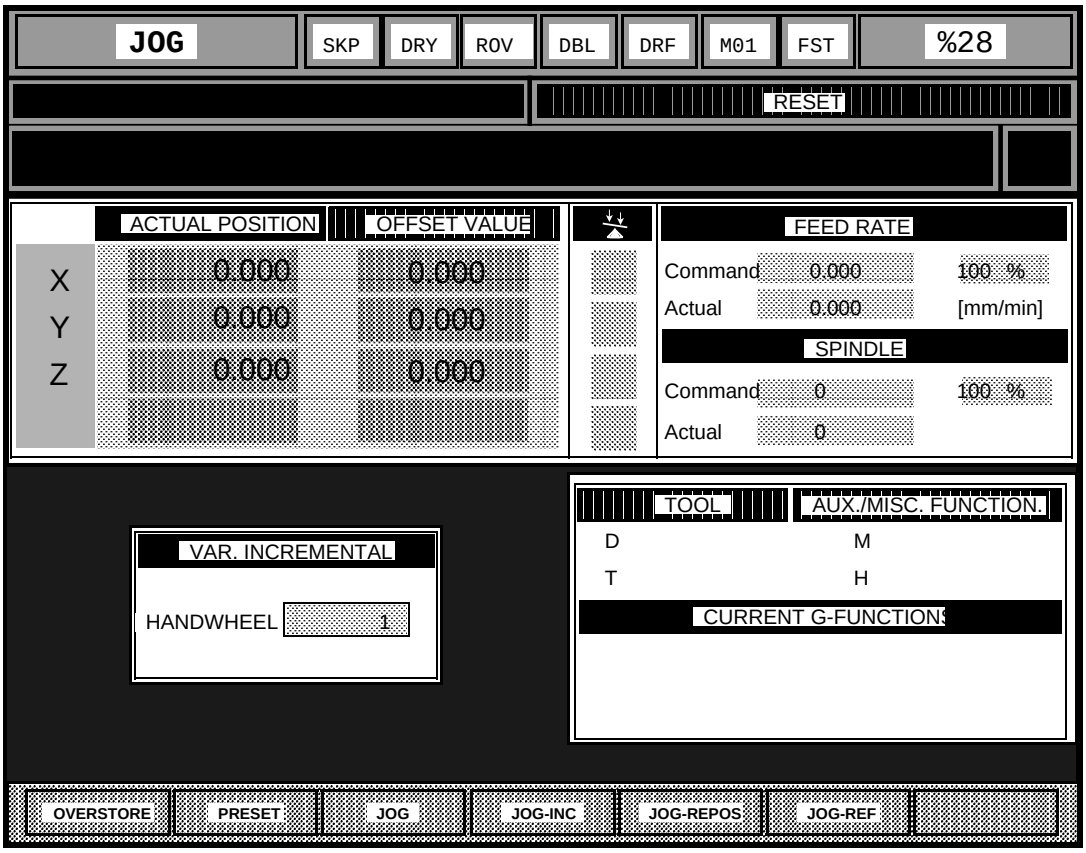
menu expansion key

VERSION softkey

The VERSION display is deselected by pressing the RECALL key.

©Siemens AG 1990 All Rights Reserved

The control is switched on and ready for operation.



Basic display for JOG mode

Switching off SINUMERIK 805

The following applies when switching off the control or the whole system:

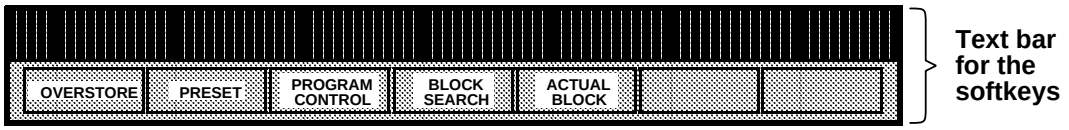
Please follow the advice
of the machine tool manufacturer!

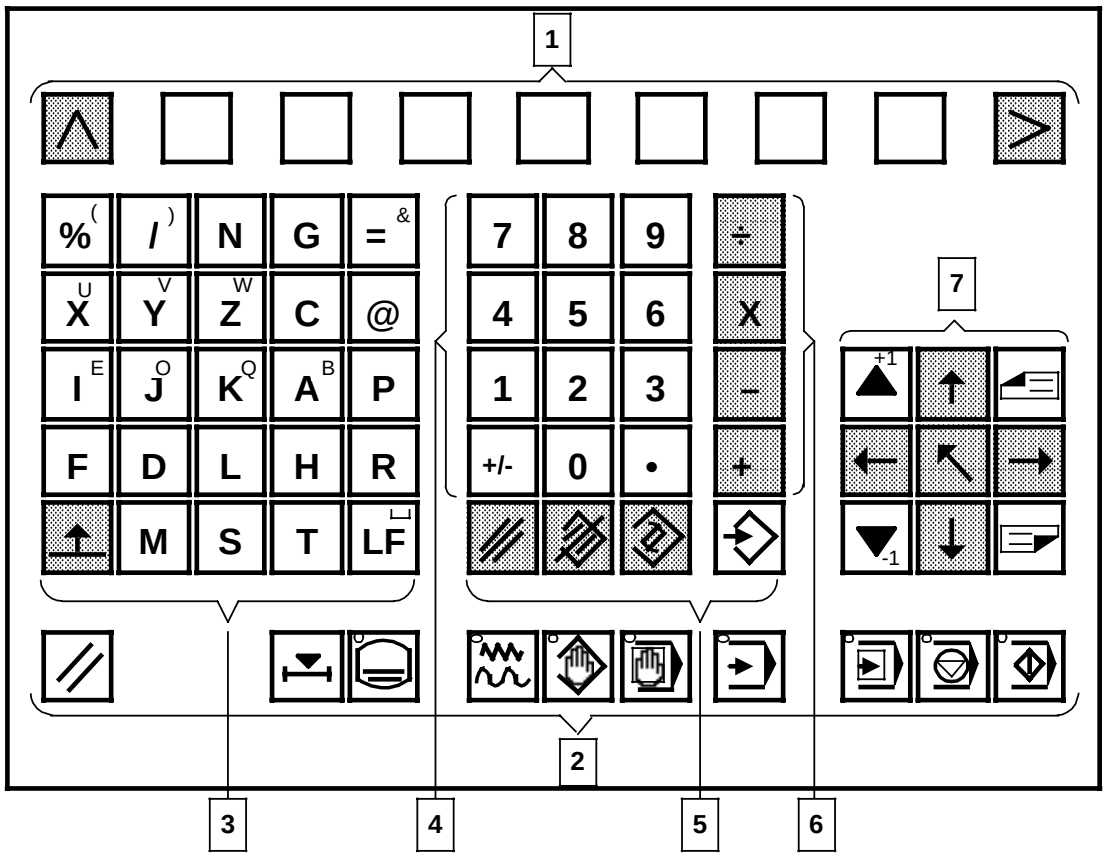
In principle it can be said that: the control is switched off by switching off the input voltage of the power supply unit of the central controller.

©Siemens AG 1990 All Rights Reserved

2 Description of Operator Controls and Areas

2.1 Operating elements of the operator keyboard





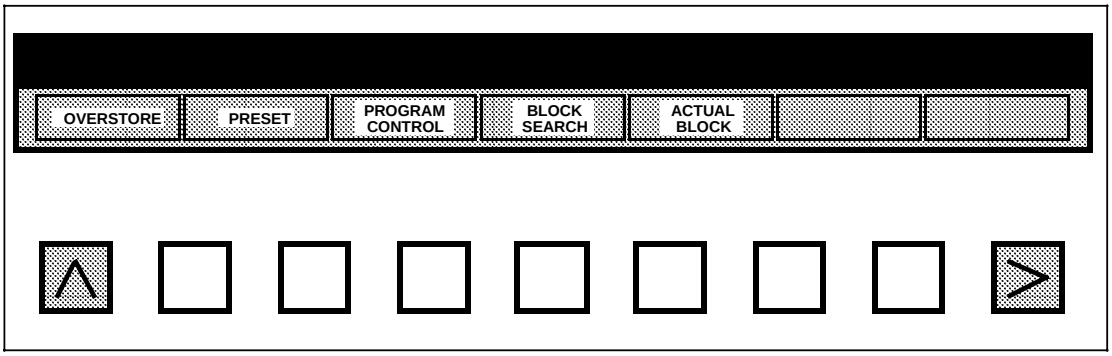
Key groups - functions:

1	Softkeys with menu control keys
2	Machine function keys with operating area key
3	Address keys
4	Numeric keys
5	Edit and input keys
6	Arithmetic keys
7	Cursor keys with decrement, increment and page keys

©Siemens AG 1990 All Rights Reserved

2.1.1 Explanation of the operating elements of the operator keyboard

Softkeys with menu control keys 1

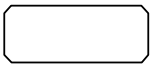


Softkeys with text bar

Softkey

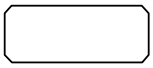
Press one of the seven softkeys (a softkey is a key without a fixed function) select the softkey function offered in the text bar.

Softkey text



In the text bar the softkey function of the softkey below is displayed.

Note:

In the text the pictogram  is used if pressing a key is necessary.

MORE key for menu control/extension of the text bar in the same menu

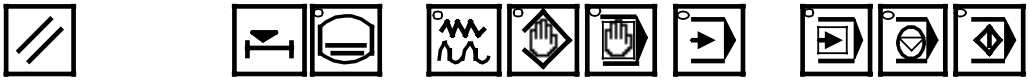
Pressing this key allows you to view additional softkey functions (if available) from the same menu in the text bar.

RECALL key for menu control/return to the text bar in the higher-level menu

- By pressing this key, you move back to the higher-level menu.
- In addition, you can also close an opened window, e.g. "INPUT VALUE TOO LARGE" or "SEARCH", with this key.

©Siemens AG 1990 All Rights Reserved

Machine function keys with operating area key **2**



RESET

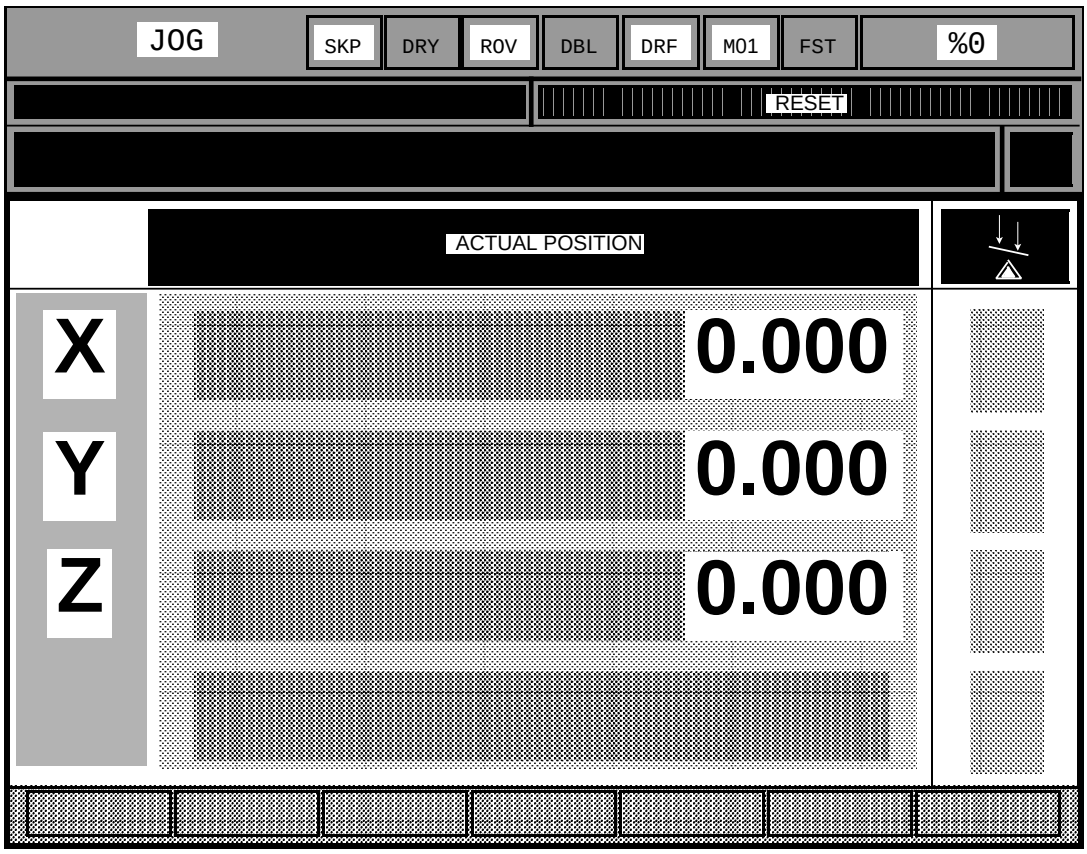
Pressing the RESET key causes the following:

- Termination of the active part program execution
- Acknowledgement (deletion) of alarms 100* to 196*, 2000 to 2999
- RESET state of the control (display on screen)
  - The axes remain synchronized with the machine.
  - The buffer and user memory are erased.
  - The control is ready to execute a new program.

©Siemens AG 1990 All Rights Reserved

Actual position with double-height characters

If you operate this key in the machine area of the control, the actual positions of the axes concerned are displayed in double-height characters.



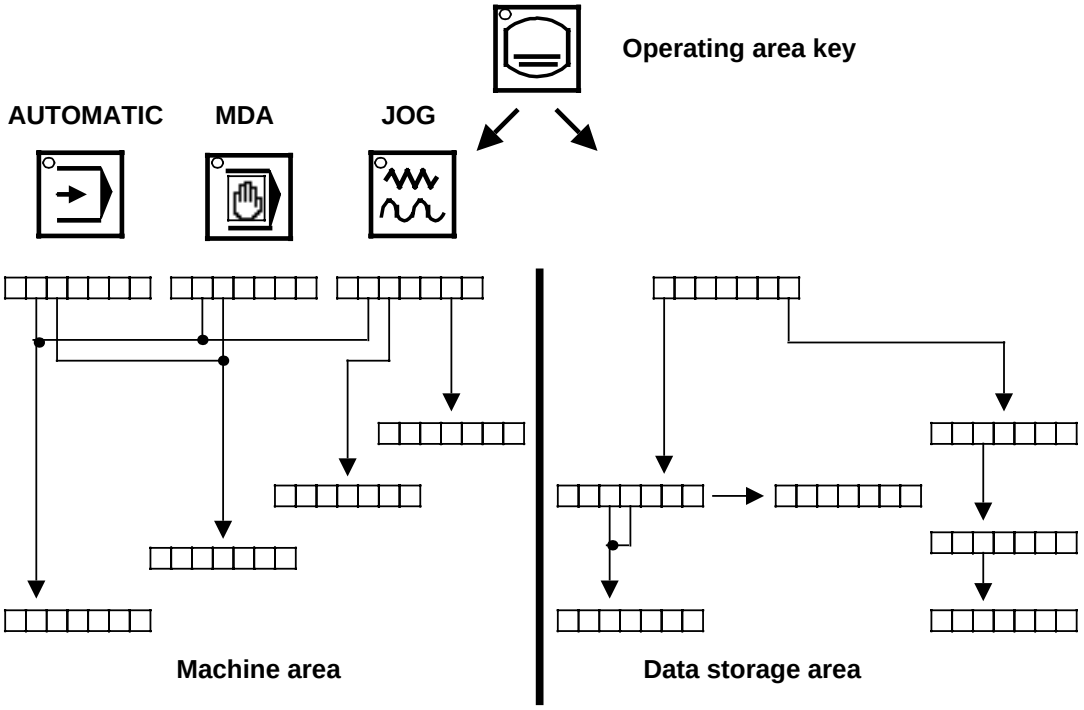
Further information from the previous display is faded out. Operating the key again returns the previously displayed menu to the screen.

©Siemens AG 1990 All Rights Reserved

Operating area key

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

Termination of active part program execution.



- You are located in the machine area with the functions which are dependent on the type of mode (LED not on):
  Using this key, it is possible to branch to the basic menu from any menu in this area.

- You are located in the data area with functions for displaying and changing data (R parameters etc.) and NC programs (LED on):
  Using this key, it is possible to branch to the basic menu of the selected mode from any menu in this area.

JOG-Taste

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 selected. The selected mode is also displayed in the working field on the screen.

©Siemens AG 1990 All Rights Reserved

TEACH-IN key

(Available soon)

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 selected. 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 selected. The selected mode is also displayed in the working field on the screen.

SINGLE BLOCK key

This key offers you the possibility of executing a part program block by block in the AUTOMATIC mode. Repeated operation of this key switches the function on and off.

- LED not on: single-block execution inactive
- LED on: single-block execution active
  - STOP: SINGLE BLOCK appears on the screen
  - When the NC start key has been pressed the active block is processed, the screen display STOP: SINGLE BLOCK disappears whilst the block is processed.
  - Once the block has been processed, the STOP: SINGLE BLOCK again appears on the screen
  - The next block is processed when the NC start key is pressed, etc.

©Siemens AG 1990 All Rights Reserved 6ZB5 410-0EE02

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. Processing of the program from the point of interruption can be reinitiated by operating the NC start key.

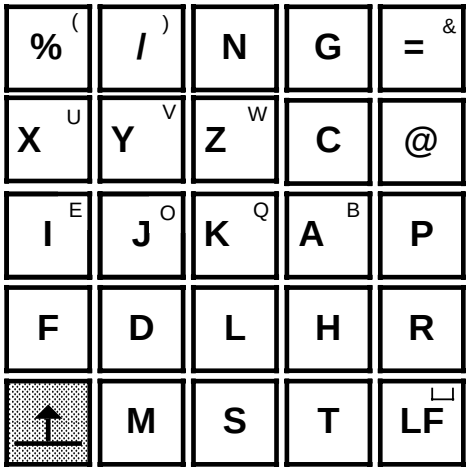
The LED and display mentioned above then go out.

NC START key

- AUTOMATIC mode:
  The part program which has been called up is initiated with the active 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:
  You press this key to start execution of the entered NC block.

©Siemens AG 1990 All Rights Reserved

Address keys **3**



Key shift

When this key is operated, the **second function** becomes active for keys with a double assignment.

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

% (

Symbol: open parenthesis

Sign for "program start"

/)

Symbol: closed parenthesis

Sign for "skip block"

N

Address for "block number": N...

G

Address for "preparatory function": G...

= &

Ampers and

Equals sign

©Siemens AG 1990 All Rights Reserved

Letter U

Address for "path information" of the X axis

Letter V

Address for "path information" of the Y axis

Letter W

Address for "path information" of the Z axis

Letter C

Address for "programm control functions"

Letter E

Address for "interpolation parameter"I

Letter O

Address for "interpolation parameter" J

Letter Q

Address for "interpolation parameter" K

Address for "radius"/letter B

Address for "angle"/letter A

Address for "subroutine pass count"

Address for "feedrate"/letter F

©Siemens AG 1990 All Rights Reserved

D	Address for "tool offset number"/letter d
L	Address for "subroutine number"
H	Address for "auxiliary functions"
R	Address for "parameters"
M	Address for "additional functions"
S	Address for "spindle speed"
T	Address for "tool number"
LF ⌐	Blank Sign for "block end" (**L**ine **F**eed)

©Siemens AG 1990 All Rights Reserved

The following characters are not marked on the keyboard, as they are only used to enter @ commands for the creation of cycles (second functions).

"a" is to be found by shift + F key 

”b” is to be found by shift + D key 

”c” is to be found by shift + L key 

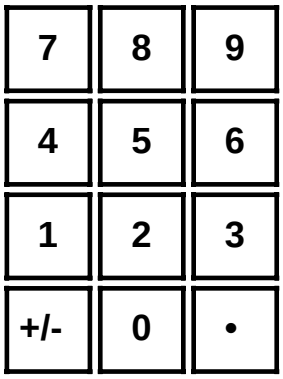
”d” is to be found by shift + H key 

”e” is to be found by shift + R key 

”f” is to be found by shift + M key 

©Siemens AG 1990 All Rights Reserved

Numeric keys **4**



Numeric keys

0 ... **9** Numbers 0 ... 9

Plus/minus key

+/-

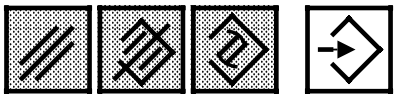
- By operating this key you reverse the sign in front of the word in the opened input field:
  \+ - / - + .
- Additionally, you can activate or deactivate functions selected with the cursor in certain menus.

Decimal point key

• Decimal point

©Siemens AG 1990 All Rights Reserved

Edit and input keys **5**



DELETE key	**Delete character**
	With this key you delete characters in the opened input field • you delete the character selected by the cursor if the key is pressed once • continuous operation of the key deletes all successive characters from the selected character towards the right.

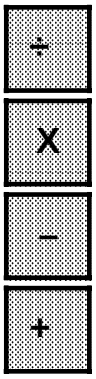
CANCEL key	**Delete input field**
	You delete the contents of the opened input field with this key.

EDIT key	**Open input field**
	• With this key you open the input field selected by the cursor. The input field must be opened before an entry can be made. • If you conclude an input with this key, the input value is not accepted and the previous value is the value in the closed input field.

Entry key	
	The input is concluded with this key: • The value in the edited input field is entered accordingly.

©Siemens AG 1990 All Rights Reserved

Arithmetic keys **6**

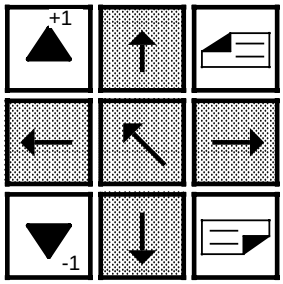


SINUMERIK 805 offers the following arithmetic functions:

÷	Division key
X	Multiplication key
–	Subtraction key
+	Addition key

©Siemens AG 1990 All Rights Reserved

Cursor keys with decrement, increment and page keys 7



Cursor left/right movement key

Using these keys, you move the cursor on the screen to the left and to the right.

Cursor up/down movement key

Using these keys, you move the cursor on the screen up and down.

Home key

Operating this key positions the cursor in the top left input field.

©Siemens AG 1990 All Rights Reserved

Increment/Decrement key

- Operation of these keys changes increment, decrement the incremental dimension of the field selected by cursor (JOG-INC or HANDWHEEL) in the JOG, JOG-INC AND JOG-REPOS modes.

  Areas: JOG-INC: 1, 10, 100, 1000, 10000 [unit/IS]

  HANDWHEEL: 1, 10, 100 [unit/IS]

- You increment/decrement the selected operands in the DIAGNOSIS window.

Page up/down key

By operating either of these keys, you can change the active screen display if further displays are available.

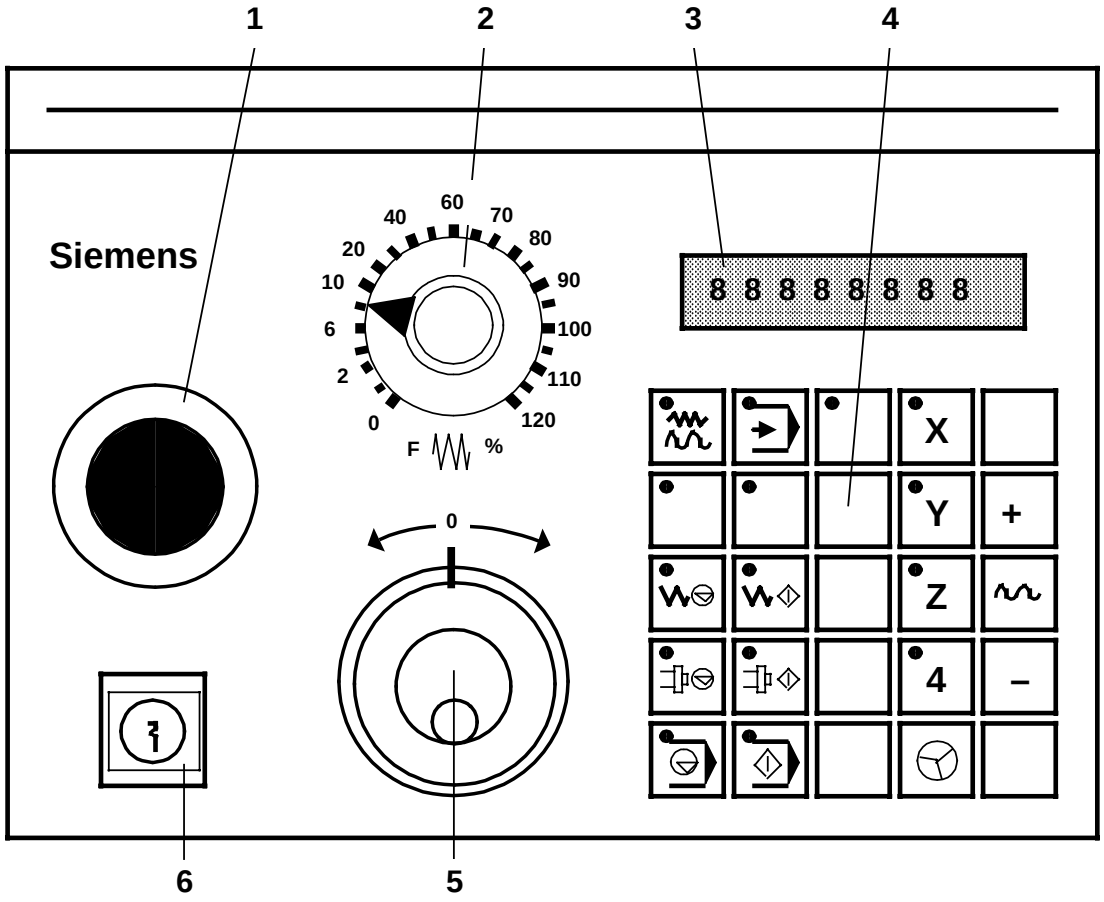
You page one display back.

You page one display forward.

©Siemens AG 1990 All Rights Reserved

2.2 Operating elements of the Hand-held unit

The description applies to the use of standard labelling strips and the corresponding programmed functions (see Interface Part1, Signals, Section 4.1, Standard PLC program).



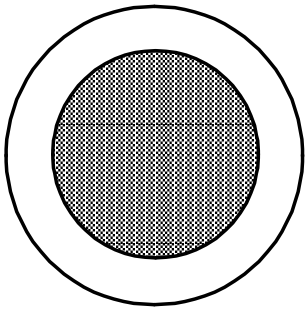
Explanation of operating elements:

1	EMERGENCY STOP switch
2	Feedrate/rapid traverse override switch
3	Axis actual position and distance to go display
4	Key group with LED display (STANDARD labelling)
5	Handwheel
6	Keyswitch

©Siemens AG 1990 All Rights Reserved 6ZB5 410-0EE02

2.2.1 Explanation of operating elements of hand-held unit

Emergency stop switch **1**



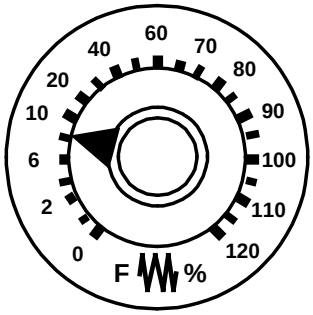
The red switch is operated in **emergency situations:**

- 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 breaking power to a defined state.

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 %.

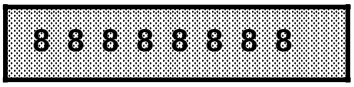
Control range: 0% . . . . . 120% of the ***programmed*** feedrate.
In rapid traverse, the 100% value is not exceeded.

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 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!

©Siemens AG 1990 All Rights Reserved

Axis actual position and distance to go display **3**



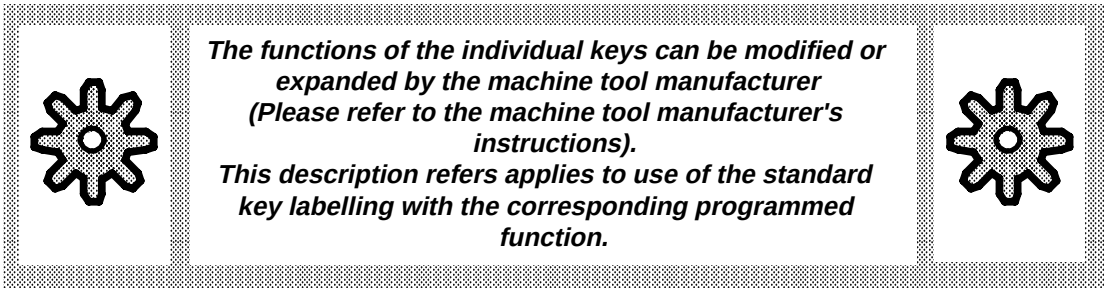
Depending on the selected axis and function, the actual position or distance to go of each axis is displayed.

Note:

The display remains dark if no axis is slected via PLC-NC interface.

Keys with LED display **4**

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



Jog key

JOG mode is selected with this key.

Automatic key

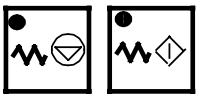
AUTOMATIC mode is selected with this key.

When JOG or AUTOMATIC modes are selected, the corresponding LED keys turn on. The selected mode is additionally displayed in the operating mode field on the screen.

The modes can also be selected with the appropriate keys on the operator keyboard.

©Siemens AG 1990 All Rights Reserved

Feedrate stop / Feedrate start keys



You operate the FEEDRATE STOP key. 

- The program being processed is stopped.
- The feed drives are brought to a controlled stop.
- The FEEDRATE START key LED goes out.
- The FEEDRATE STOP key LED is on.
- The status display for FEEDRATE STOP (FST) is displayed in reverse video.

Examples for the use of FEEDRATE STOP

- when a fault is discovered whilst processing a block during operation in MDA mode,
- in JOG, JOG-INC and JOG-REPOS modes, e.g. during return to the contour,
- to change a tool,
- to input S, T, H and M functions during setting up (overstore).

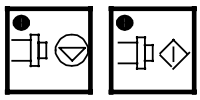
You operate the FEEDRATE START key. 

- the part program continues in the active block,
- the feedrate is increased to the value given in the program,
- the LED of the FEEDRATE STOP key goes out.
- the LED of the FEEDRATE START key is on.

The following values are specified in the 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.
(See the machine tool manufcturer's documentation.)

©Siemens AG 1990 All Rights Reserved

Spindle Stop / Spindle Start keys



You operate the SPINDLE STOP key. 

- The spindle is brought to a standstill.
- The LED of the SPINDLE START key goes out.
- The LED of the SPINDLE STOP key is on.

Example for the use of SPINDLE STOP

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

You operate the SPINDLE START key. 

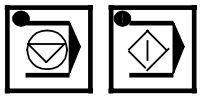
- the spindle speed is accelerated to the value specified in the program.
- the LED of the SPINDLE STOP key goes out.
- the LED of the SPINDLE START key is on.

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)

©Siemens AG 1990 All Rights Reserved

NC stop / NC start keys



You operate the NC-STOP key. 

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

You operate the NC START key in the selected mode AUTOMATIC. 

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

NC STOP and NC START commands can also be given from the operator keyboard.

Axis selection keys

The required axis is selected with these keys:

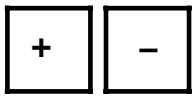
- The actual position of the selected axis is given in the axis actual position display.

- One of the four key LEDs lights up, showing the selected axis.

- After switching on, the axis is selected.

©Siemens AG 1990 All Rights Reserved

Axis direction key



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

- JOG mode selected:
  You traverse the selected axis in the required direction.

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

Rapid traverse override key

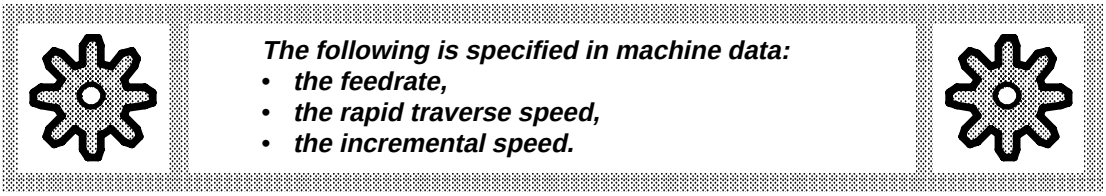
If you press the RAPID TRAVERSE OVERRIDE key

together with the 

or 

key.

The selected axis is traversed at "rapid traverse" in the appropriate direction.



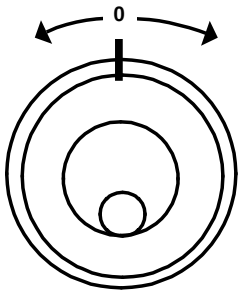
The following is specified in machine data:
- ***the feedrate,***
- ***the rapid traverse speed,***
- ***the incremental speed.***

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 effects the selected axis.

©Siemens AG 1990 All Rights Reserved

Handwheel **5**



You operate the HANDWHEEL key (key LED lights up). Using the handwheel, you can now traverse the selected axis incrementally in JOG mode or JOG INC or JOG REPOS submodes.

Keyswitch **6**

The input of data, number of functions etc. can be be protected using the keyswitch.

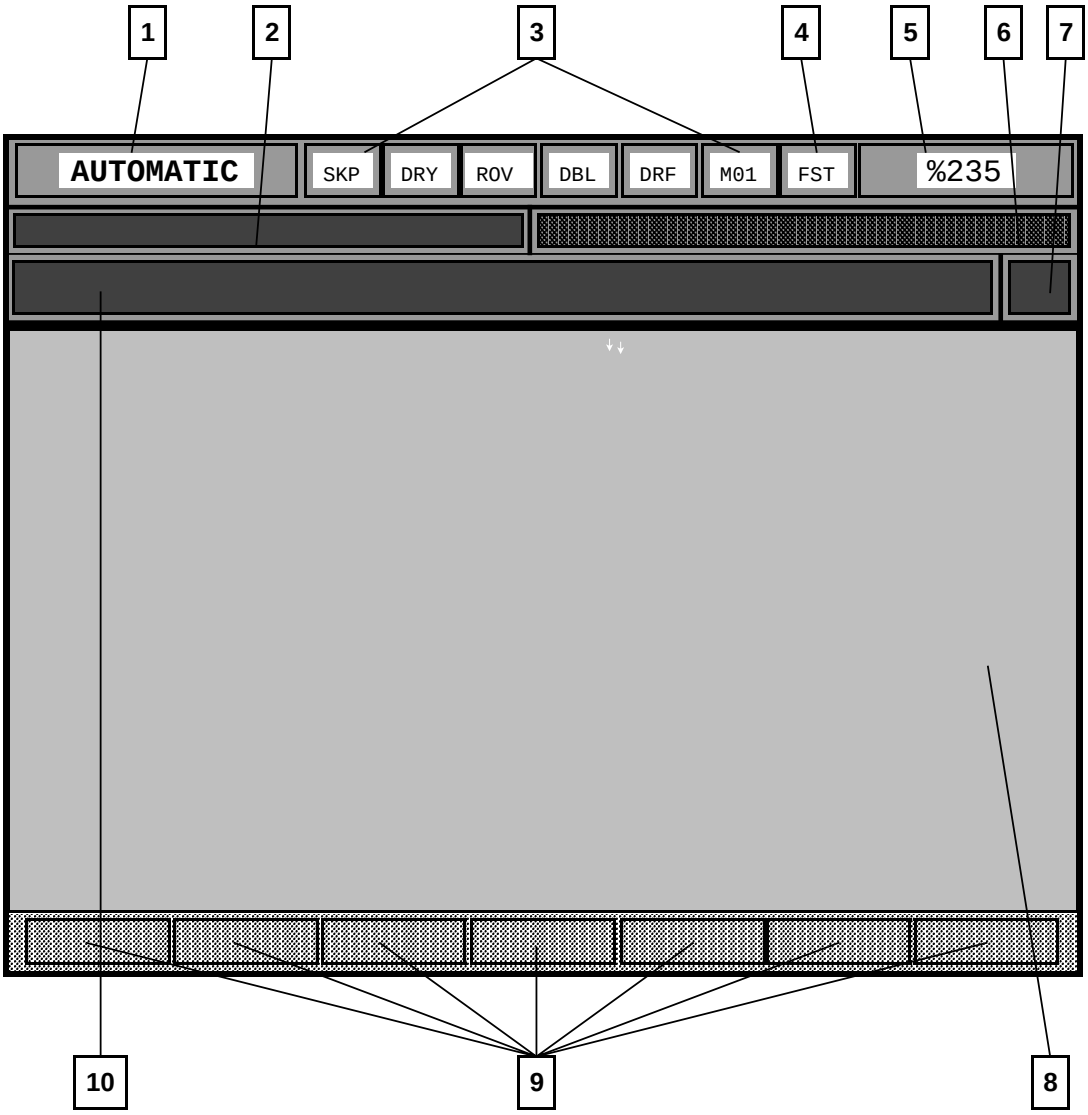
The key can be removed. All disabled functions can no longer be carried out.

The key **cannot** be removed. Disable no longer effective.

Whether the input disable in your control is active or not, depends on the machine data (see the machine tool manufacturer's documentation).

©Siemens AG 1990 All Rights Reserved

2.3 Division of the screen, softkeys



SINUMERIK 805 uses a 12" monochrome screen with a resolution of 640 x 480 pixels. The following table gives an explanation of the display functions of each of the fields:

1	Operating mode field, displays selected mode or submode.

©Siemens AG 1990 All Rights Reserved

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 for (light background ≙ active):

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 display

6 Display field for operating status

- RESET
- Program running
- Program interrupted

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 and PLC alarms and messages
(only the alarm with highest priority is displayed ≙ alarm with the lowest number).

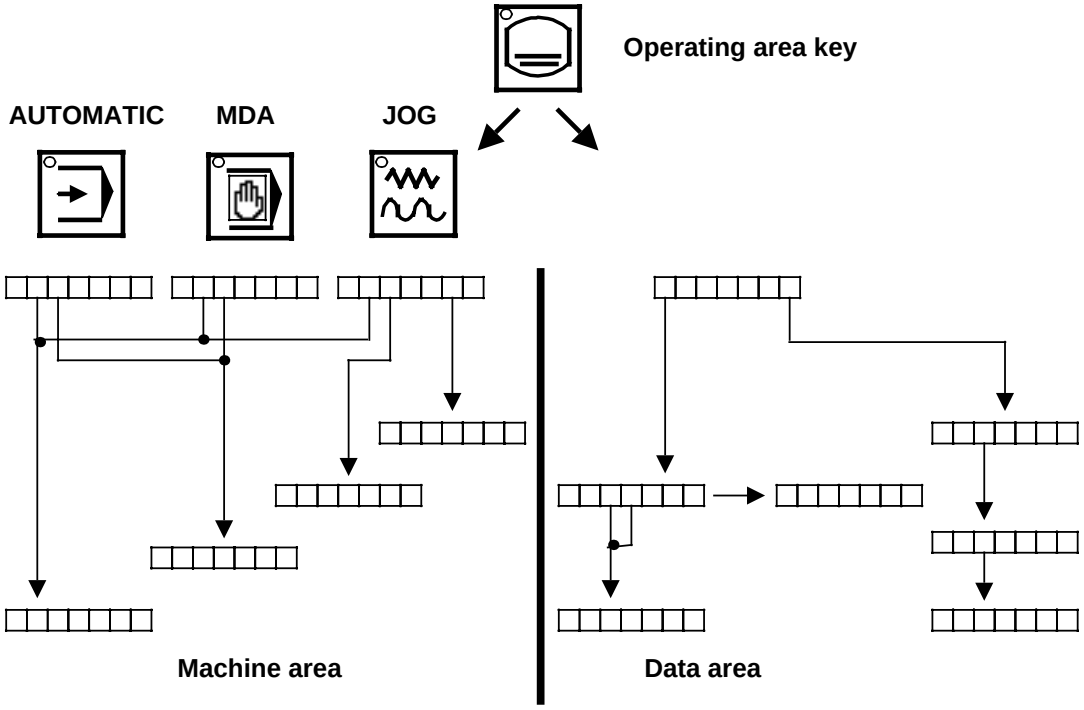
©Siemens AG 1990 All Rights Reserved

2.4 Operating area, menu tree

Operation of the SINUMERIK 805 is divided into two areas, the machine area with functions which are dependent on the type of mode and the data storage area sporting functions for data and NC program display and modification.

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

Menu trees with several branches result.



Using the operating area key, it is possible to branch to the data area **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, using the operating area key.

Comments:

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

©Siemens AG 1990 All Rights Reserved

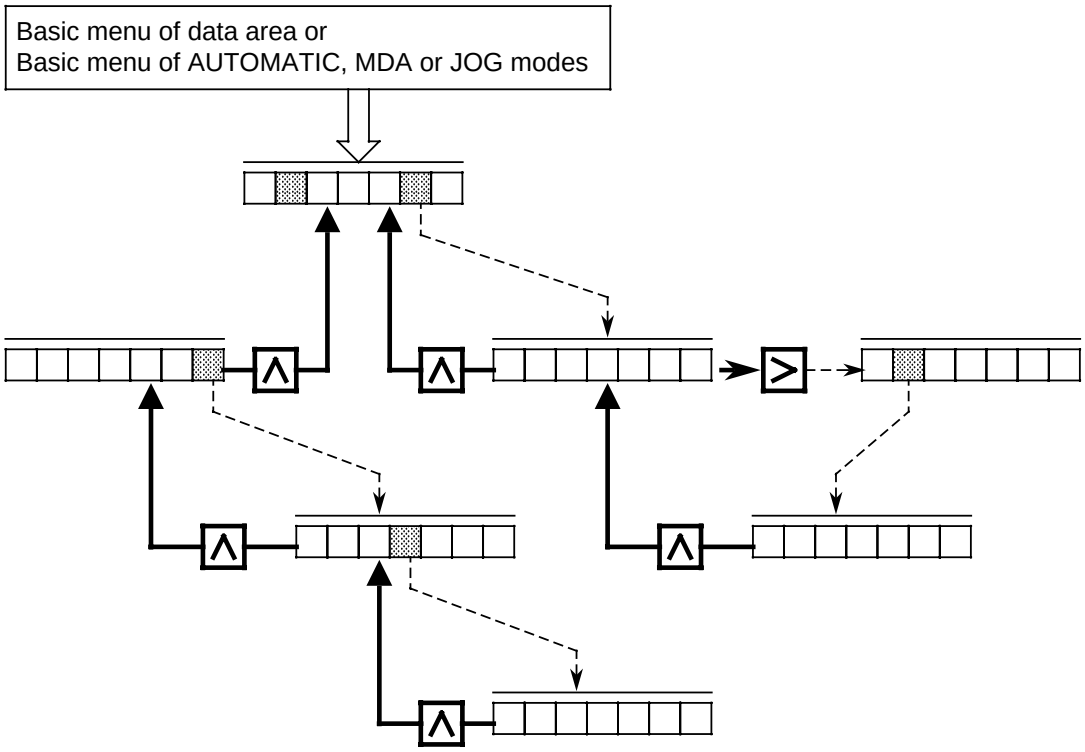
2.4.1 Branching within a menu tree

You wish to return to the higher-level menu after repeated branching.

You operate this key:

After operating this key once, the next immediate higher-level menu is displayed along with operating functions in the text bar.

- Basic menu of data area
- Basic menu of AUTOMATIC, MDA or JOG modes



You wish to refer to additional operating functions in the same menu (only if additional text bar is available).

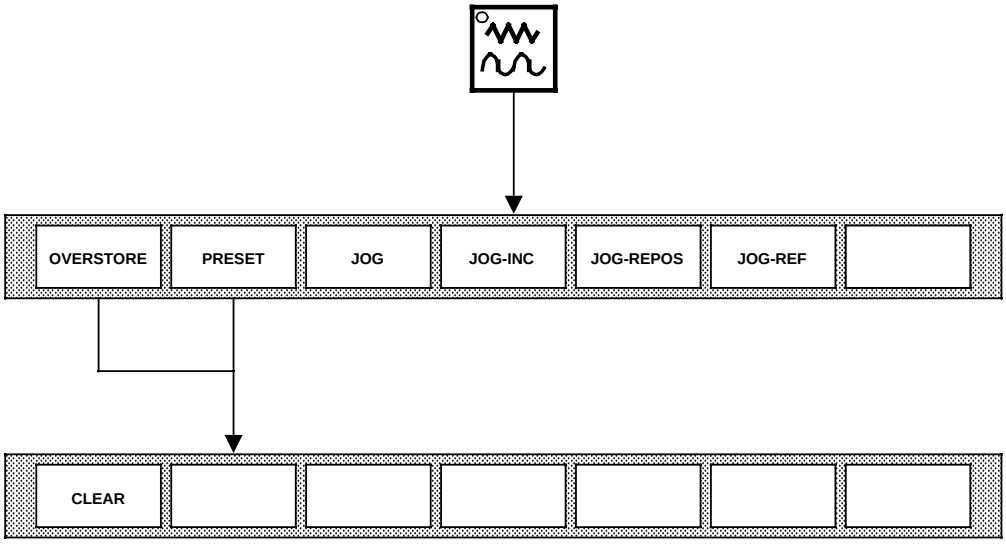
You operate the following key:

©Siemens AG 1990 All Rights Reserved

2.4.2 Machine area menu tree

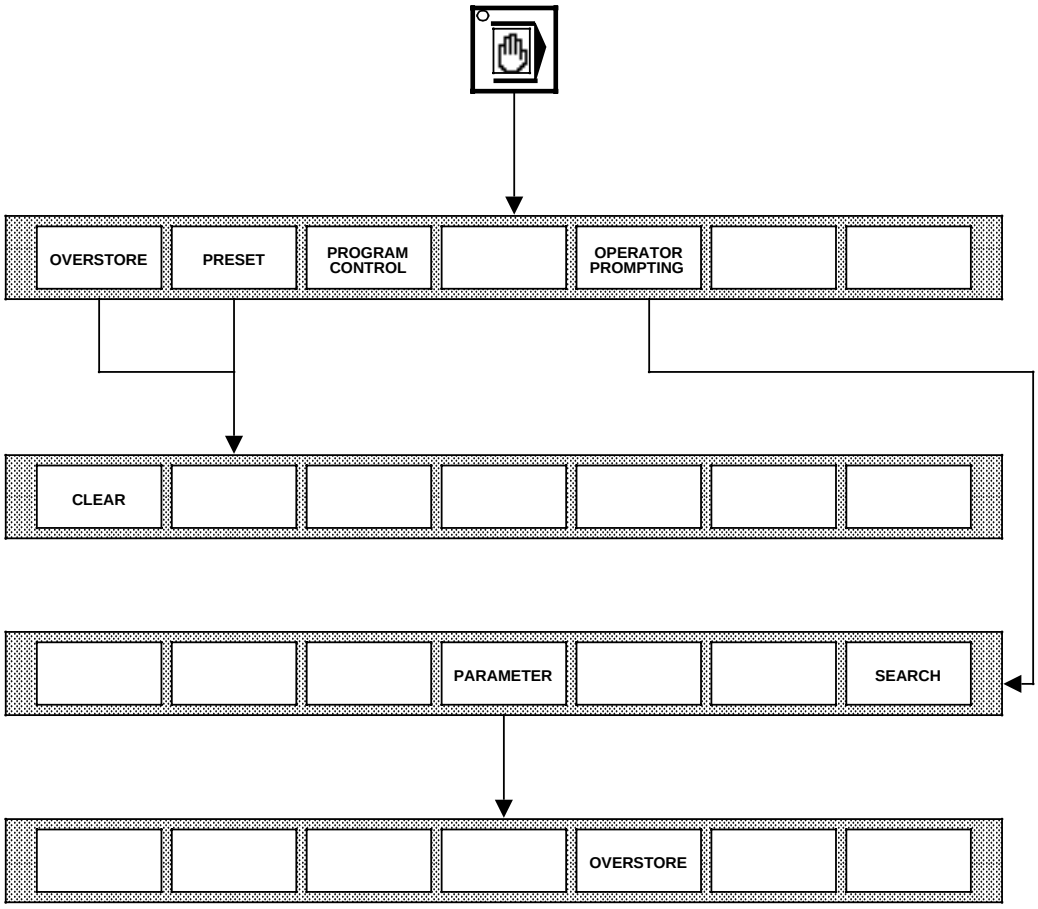
The machine area menu tree is subdivided into menu trees for JOG, AUTOMATIC and MDA modes.

2.4.2.1 JOG mode menu tree



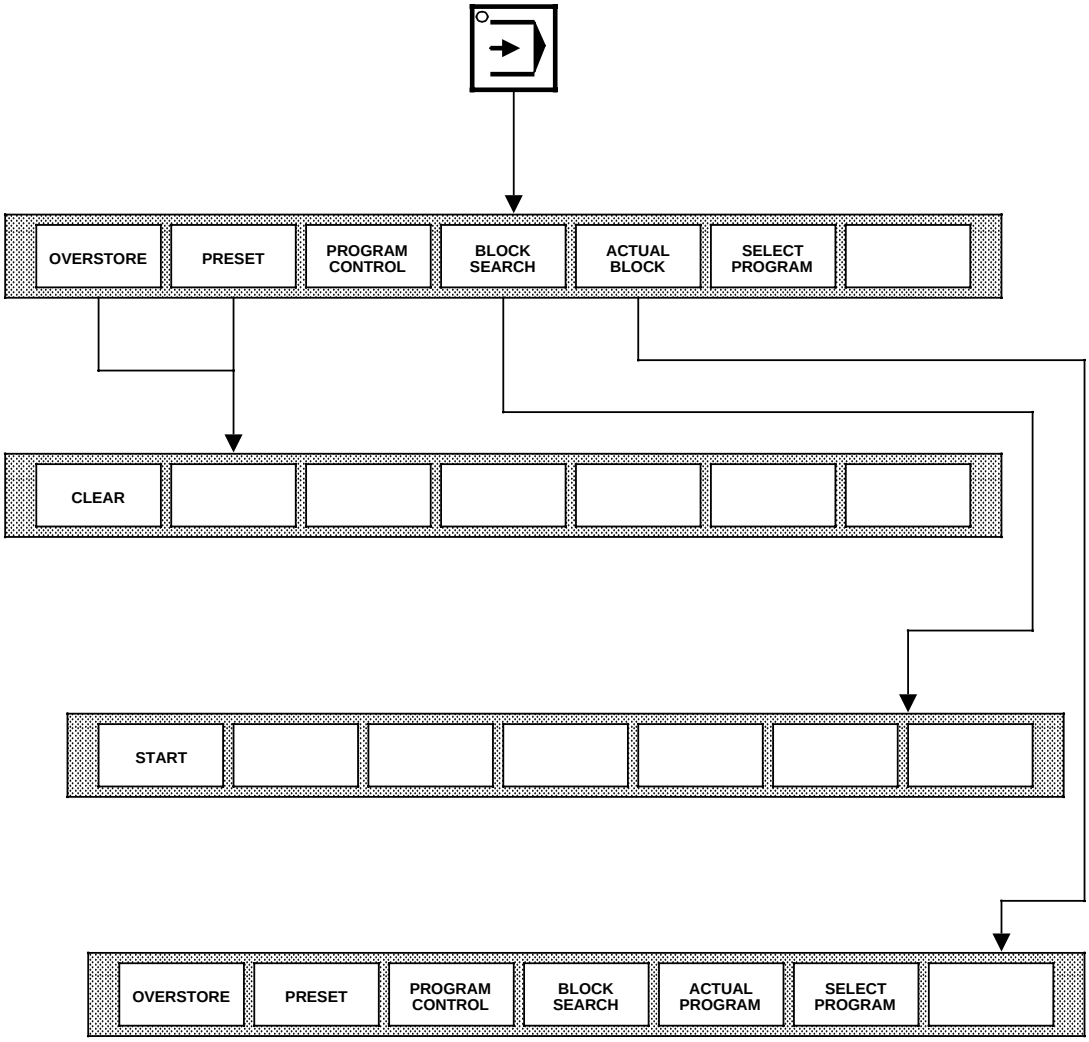
©Siemens AG 1990 All Rights Reserved

2.4.2.2 MDA mode menu tree



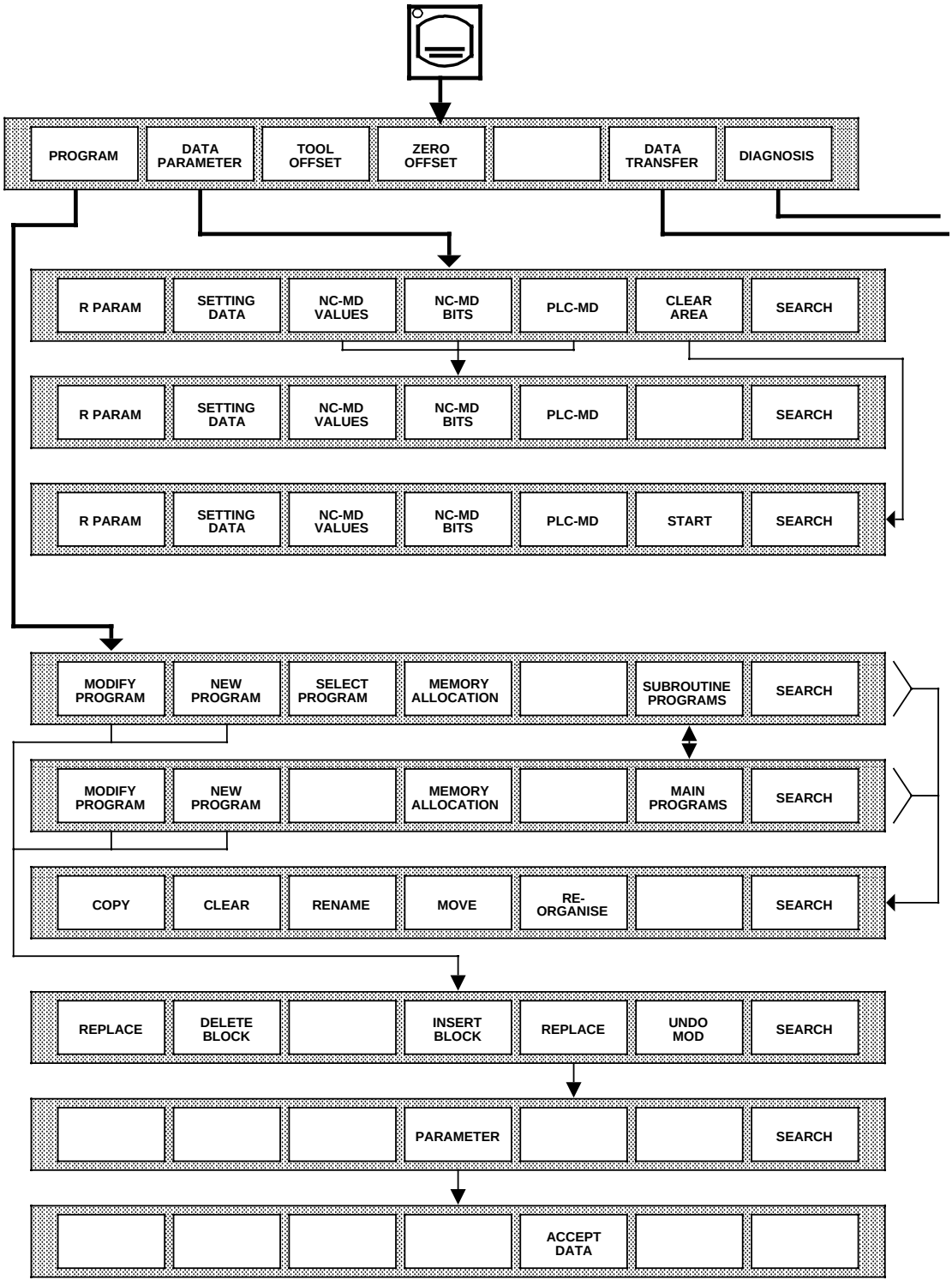
©Siemens AG 1990 All Rights Reserved

2.4.2.3 AUTOMATIC mode menu tree

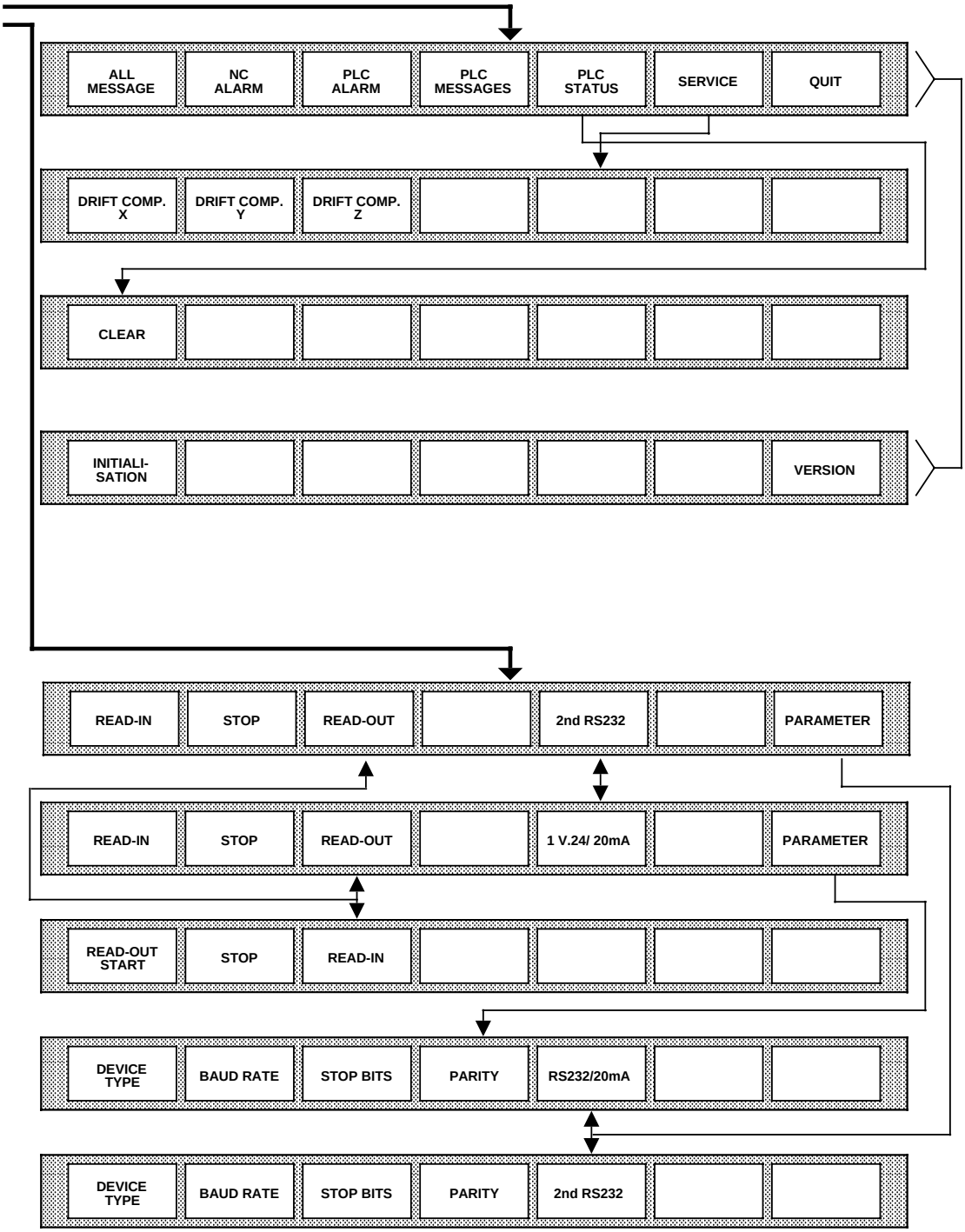


©Siemens AG 1990 All Rights Reserved

2.4.3 Data storage area menu tree



©Siemens AG 1990 All Rights Reserved

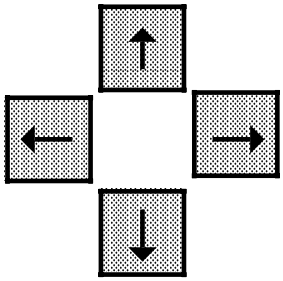


©Siemens AG 1990 All Rights Reserved

2.4.4 Entries in Input fields

The input field of the SINUMERIK 805 is marked dark on the screen. You can enter and/or change values in the input fields by operating in the following sequence:

Sequence of operation when modifying values in the input field



Using the cursor keys, you move the dark input field to the desired position within a selected menu.

Operation of the EDIT key opens the input field for the input of a value or modification of the original value.

Using the cursor keys, you can select any of the numbers in the opened input field.

Operate CHARACTER DELETE key to delete a character.

Operate INPUT FIELD DELETE to delete the whole input field.

Key in the desired value.

You enter the keyed-in value with the INPUT key.

Note:

By operating the EDIT key when the input field is open, you close the input field. The original value is maintained.

©Siemens AG 1990 All Rights Reserved

2.4.5 Calculator function

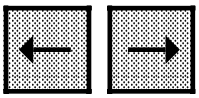
The calculator function in the SINUMERIK 805 can be applied to all input fields. It features the following arithmetical functions:

- division
- multiplication
- subtraction and
- addition

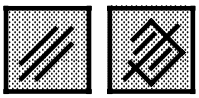
Arithmetic operation sequence

Prerequisite: You wish to execute an arithmetic operation with the value entered in the input field by means of cursor, page key, or the SEARCH softkey.

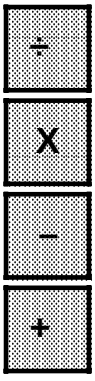
Press EDIT to open the selected input field.
The value in the field is the 1st operand. You can modify the 1st operand via the numeric keyboard.



You use the cursor keys and



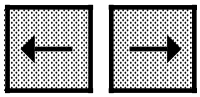
edit keys to edit.



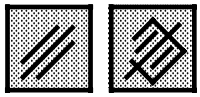
You select one of the four arithmetical keys depending on the desired operation.

©Siemens AG 1990 All Rights Reserved

Now key in the 2nd operand using the numeric keys.



You use the cursor and



edit keys to edit.

To execute the arithmetical operation, press the INPUT key.
The result is given in the closed input field.

Notes:

- The sign before the operand is reversed by operating the PLUS/MINUS key, + - and - + after entering the amount.
- The arithmetical operation is also executed by pressing the EQUALS sign. The result is given in the open input field and is the 1st operand for the next arithmetical operation.
- If you close the opened input field using the EDIT key, the original value is maintained.

©Siemens AG 1990 All Rights Reserved

3 Operating Mode Description

3.1 General

On a machine tool an NC controls

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

by means of a set program (part program).

Further preparations have to be carried out to the 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 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

3.1.1 Operating modes - overview

The SINUMERIK 805 offers 4 operating modes to bring the control into the desired state.

The following operating modes can be selected directly by operating the relevant key on the operator keyboard:

- JOG (manual operation)
- TEACH IN (available soon)
- MDA (manual input - automatic)
- AUTOMATIC

Additionally, the following submodes can be selected in JOG mode using the softkeys:

- JOG-INC (increment)
- JOG-REPOS (repositioning)
- JOG-REF (approach reference point)

The SINUMERIK 805 modes and submodes are therefore:

- JOG
- JOG-INC
- JOG-REPOS
- JOG-REF
- TEACH IN
- MDA
- AUTOMATIC

©Siemens AG 1990 All Rights Reserved

3.1.2 Operating mode selection

In the case of SINUMERIK 805, operating modes can be selected in the machine area and data area.

In this section it is assumed that the machine area has been selected.

The selected mode is displayed in the operating mode field.

To select one of the following modes:

- JOG
- TEACH-IN (available soon)
- MDA
- and AUTOMATIC

select one of the following keys on the operator keyboard:

JOG

TEACH-IN (available soon)

MDA

AUTOMATIC

The key LED lights up on the operating mode key selected. The basic display of the selected mode appears on the screen.

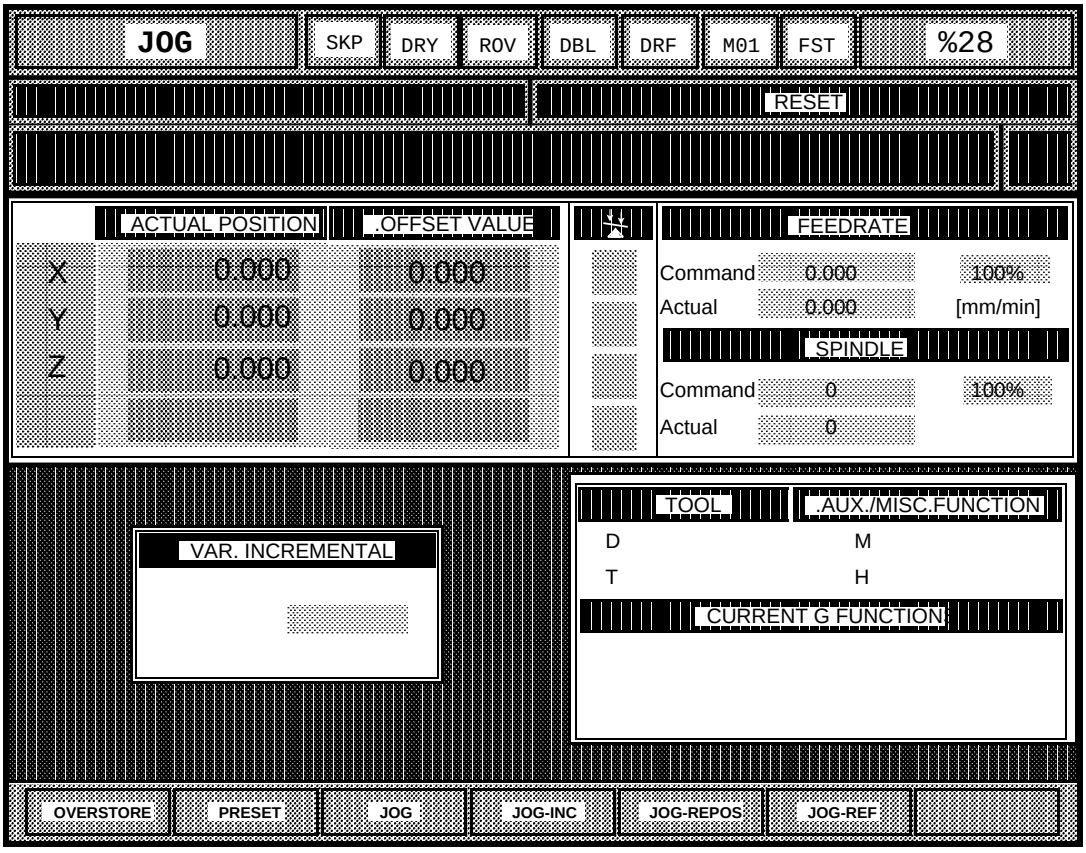
©Siemens AG 1990 All Rights Reserved

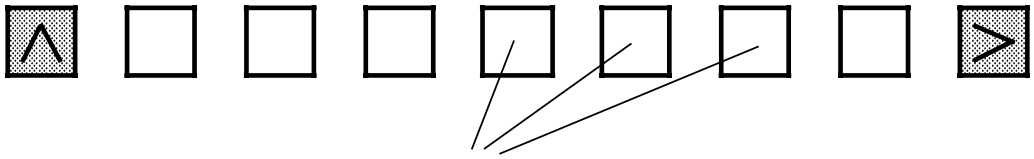
To select the submodes

- JOG-INC
- JOG-REPOS
- JOG-REF

you operate this key:

In the JOG mode basic display you can then select the desired submode by operating one of the following softkeys:





You operate one of these softkeys.

Mode JOG-INC, JOG-REPOS or JOG-REF appears on the screen.
You return to the JOG mode basic display by operating the JOG softkey.

©Siemens AG 1990 All Rights Reserved

3.1.3 RESET with change of operating mode

When changing from one selected operating mode to another, a RESET can be generated by the control, depending on the selected operating mode. The following table shows for which operating mode changes a RESET is generated:

	MDA	AUTO-MATIC	JOG	JOG REPOS	JOG INC	JOG REF
MDA		+	+	+	+	+
AUTO-MATIC	+		O	O	O	+
JOG	+	O		O	O	+
JOG REPOS	+	O	O		O	+
JOG INC	+	O	O	O		+
JOG REF	+	+	+	+	+	

RESET with change of operating mode

Notes:

- + ≙ RESET is generated
- O ≙ RESET is not generated
- The RESET generated by the control when changing the operating mode has the same effect as operating the RESET key on the operator keyboard.

Examples:

1. When changing from AUTOMATIC to JOG mode, no RESET is generated by the control.
2. When changing from JOG to JOG REF, a RESET is generated by the control.

©Siemens AG 1990 All Rights Reserved

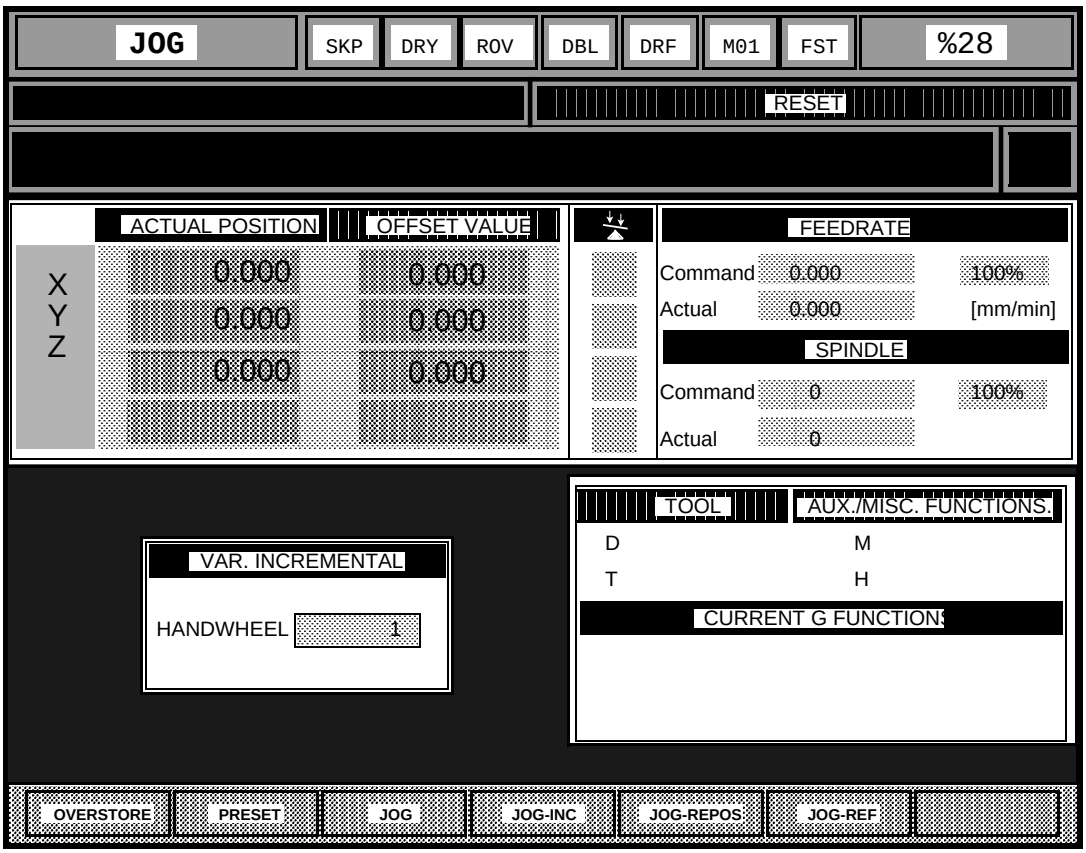
3.2 JOG mode

In this mode the individual axes can be traversed using the direction keys and the set feedrate value.
After a program interruption in the AUTOMATIC mode, the path traversed by each axis in the JOG mode referred to the point of interruption, is displayed in the OFFSET VALUE column.

3.2.1 JOG basic display

After switching on or after selection of the JOG mode in the machine area by means of this key:

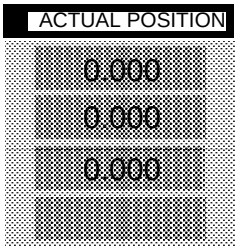
the following display appears:



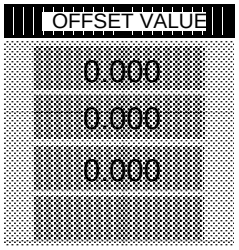
©Siemens AG 1990 All Rights Reserved

3.2.2 Notes on JOG basic display

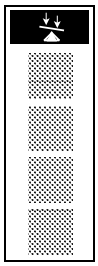
Display of addresses of each of the axes. After an axis has been referenced, the appropriate field is darkened.



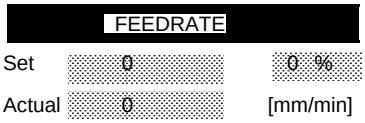
The actual position of each of the axes is shown in these fields.



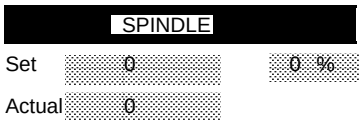
After interruption of a program in AUTOMATIC mode using the NC-STOP key, the path traversed by each axis in JOG mode referred to the point of interruption is displayed here.



Movement of an axis in a positive or negative direction is indicated by a plus or minus sign in the appropriate field.

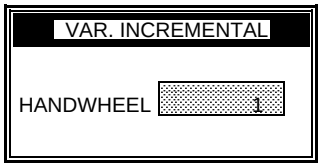


Display of the command and actual value for the feed rate and type of feed rate and position of the feedrate/rapid traverse override switch.

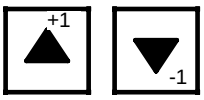


Display of spindle speed command and actual value and spindle override switch position.

©Siemens AG 1990 All Rights Reserved 6ZB5 410-0EE02



This value will be the weighting for the handwheel increment value (possible values: 1, 10, 100) when the axis is traversed using the handwheel.This value can be modified by operating the following keys:



Increment/Decrement key

Short description of the softkey functions:

- OVERSTORE : Entry of auxiliary functions (s. Section 5.3.12)
- PRESET : Set actual value (s. Section 5.3.13)
- JOG : Select JOG mode (s. Section 3.2.1)
- JOG-INC : Select JOG-INC submode (s. Section 3.3)
- JOG-REPOS : Select JOG-REPOS submode (s. Section 3.4)
- JOG-REF : Select JOG-REF submode (s. Section 3.5)

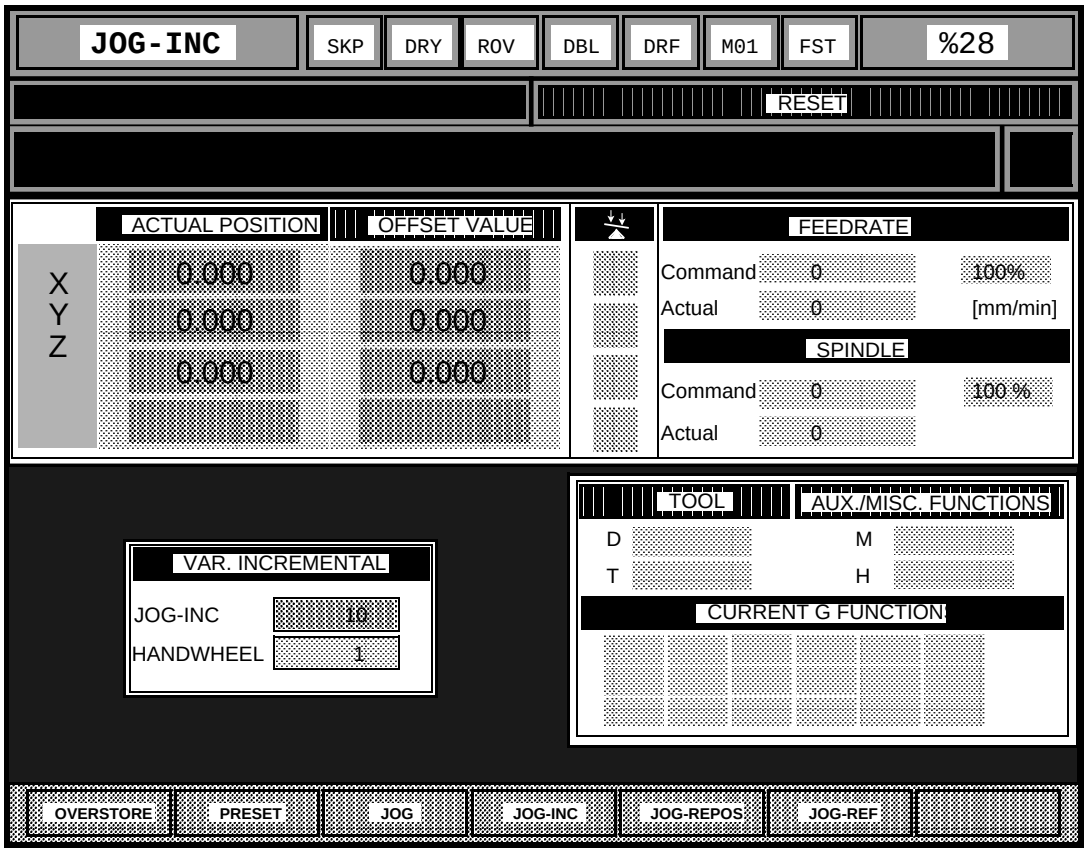
©Siemens AG 1990 All Rights Reserved

3.3 JOG-INC submode

In this case, machining is carried out with the NC in the jog mode (by hand). Each time you operate the direction key, the selected axis is traversed by the set increments in the appropriate direction.

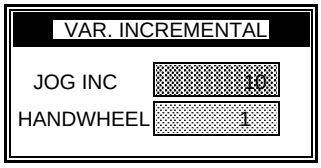
3.3.1 JOG-INC basic display

When you are located in the JOG basic display, you can select the JOG-INC submode by operating the JOG-INC softkey. After selection, the following basic display appears:



©Siemens AG 1990 All Rights Reserved

3.3.2 Notes to JOG-INC basic display



If an axis is traversed using the +/- key on the hand-held unit or using the handwheel, the selected axis is traversed by the set number of increments each time the key is pressed or for each handwheel pulse.
Possible values:

JOG-INC: 1, 10, 100, 1000, 10000 µm or variable between 0 and 10000.

Handwheel: 1, 10, 100 µm

Modifications to VAR. INCREMENTAL weighting:

JOG INC input field:

1) You enter any value between 1 and 10000, as described in Section 2.4.4 (Input field entries).

2) You operate key 

or 

to select the five values (1, 10, 100, 10000) in ascending or descending order.

HANDWHEEL input field: You operate key 

or 

to select the three values (1, 10, 100) in ascending or descending order.

Explanations to all other fields: S. Section 3.2.2.

©Siemens AG 1990 All Rights Reserved

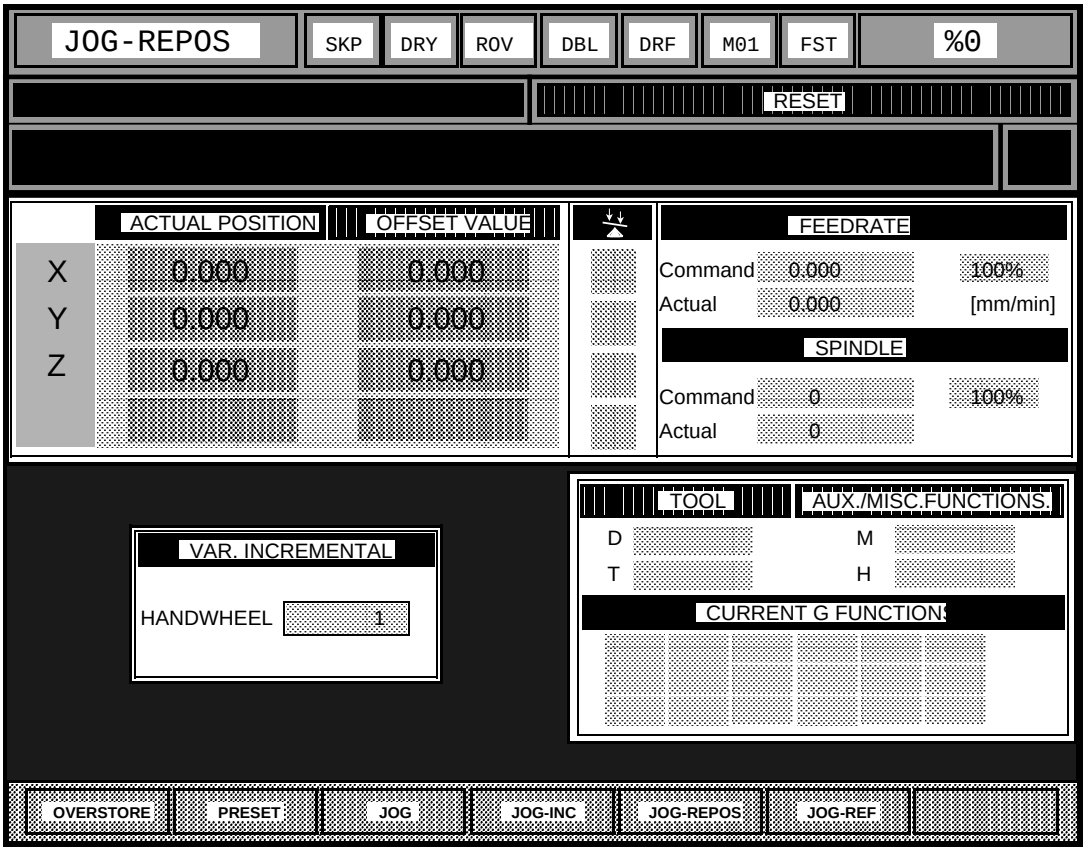
3.4 JOG-REPOS submode

After a program interruption, you can move away from the contour, for example, by switching to JOG mode. In the JOG-REPOS mode you move the tool back to the point of interruption by means of the direction keys and the set feedrate value.

The OFFSET VALUE 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 "OFFSET VALUE" display is zero and the direction keys are no longer active.

3.4.1 JOG-REPOS basic display

The following display appears after you have operated the JOG-REPOS softkey in the JOG mode basic menu:



3.4.2 Notes on JOG-REPOS basic display

The explanation of the fields given in Section 3.2.2 applies here.

©Siemens AG 1990 All Rights Reserved

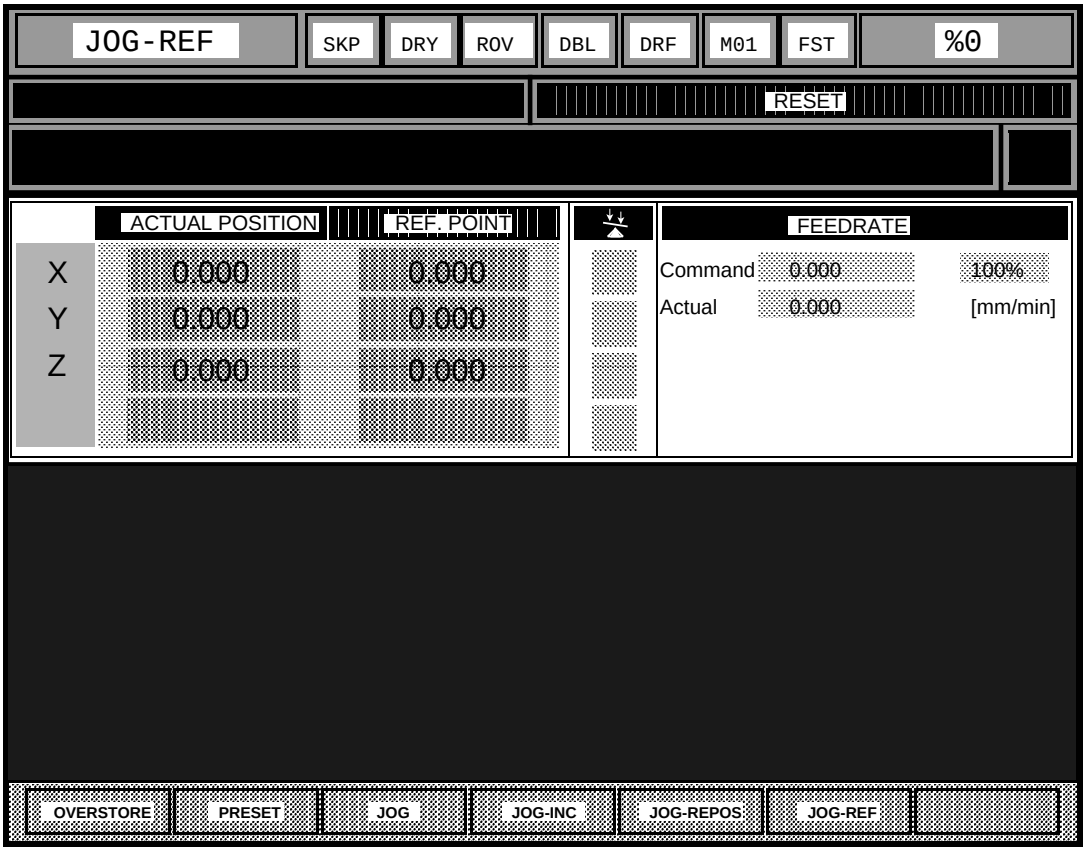
3.5 JOG-REF submode

The control and the machine are synchronized in this mode after the control has been switched on.The reference point is approached with each of the axes, one after the other.

When the reference point is reached, the reference point value for the axis concerned stored in the machine data is displayed as the actual position, and the corresponding axis name field is darkened.

3.5.1 JOG-REF basic display

You are located in the JOG mode basic menu and select JOG REF mode by operating the softkey concerned. The following display appears:



3.5.2 Notes on JOG REF basic display

The explanation of the fields given in Section 3.2.2 applies here.

©Siemens AG 1990 All Rights Reserved

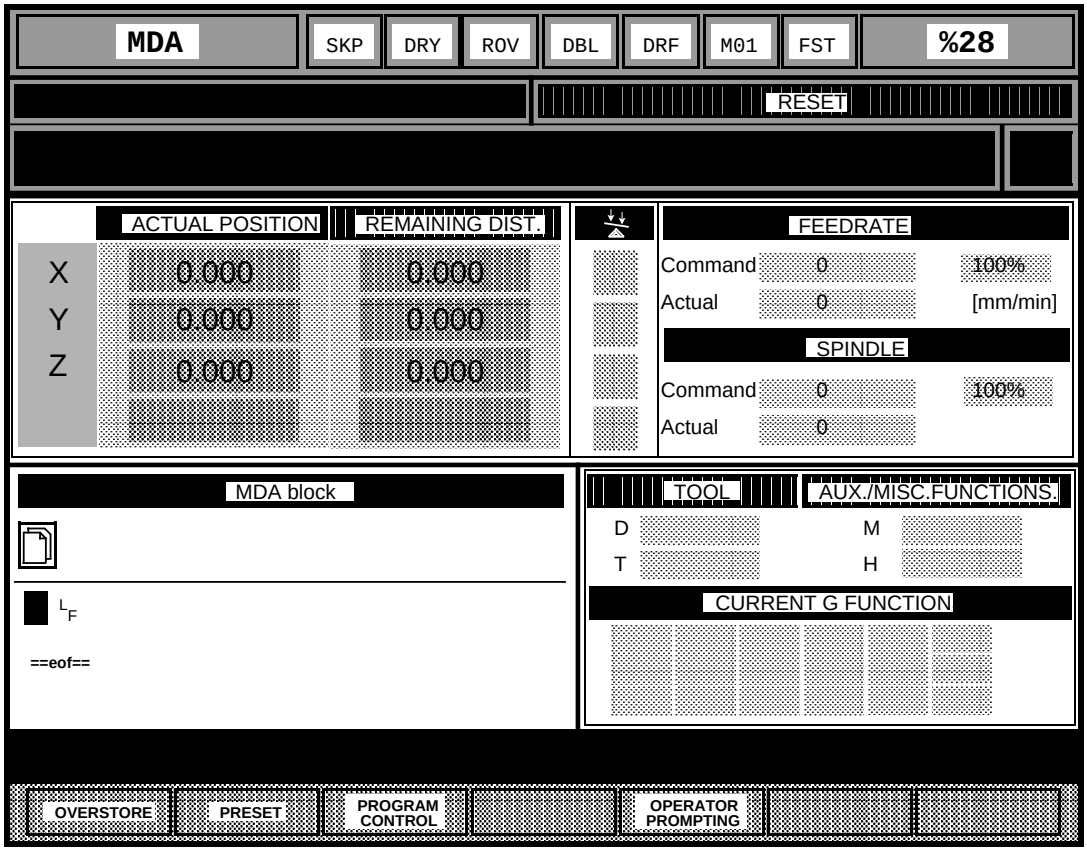
3.6 MDA mode

It is in this mode that the individual part program blocks can be entered in the buffer storage of the control using the operator keyboard. The control processes the entered block when the NC START key is pressed and clears the buffer memory for new entries.

3.6.1 MDA basic display

After selecting the MDA mode in the machine area by means of the following key:

the following basic display appears:



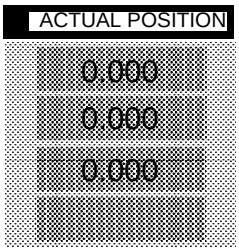
Short explanation of softkeys:

- OVERSTORE : Entry of auxiliary functions (s. Section 5.3.12)
- PRESET : Set actual value (s. Section 5.3.13)
- PROGRAM CONTROL : Modify sequence of a part program (s. Section 5.3.5)
- OPERATOR PROMPTING : Parameter support for subroutine programs (s. Section 5.3.11.1)

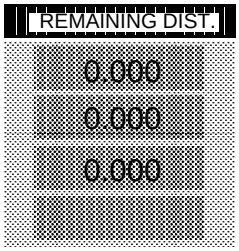
©Siemens AG 1990 All Rights Reserved 6ZB5 410-0EE02

3.6.2 Notes on MDA basic display

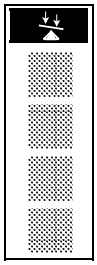
Display of the addresses of each of the axes. After the reference point of an axis has been set, the appropriate field is darkened.



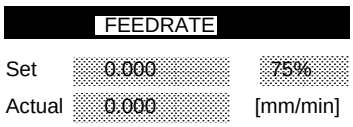
The actual position of each of the axes is shown in these fields.



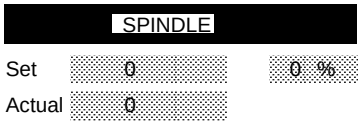
The distance to go produced during the processing of the commenced NC block, is displayed here.



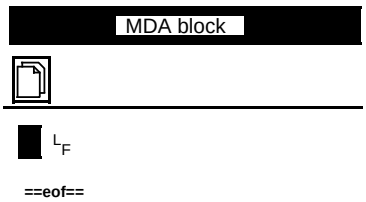
Movement of an axis in a positive or negative direction is indicated by a plus or minus sign in the appropriate field.



Display of the feedrate set and actual value, type of feedrate and position of the feedrate/rapid traverse override switch.



Display of spindle speed set as well as position of spindle speed override switch.



You enter the NC block to be processed in this input field. After the block has been processed, this field is cleared.

©Siemens AG 1990 All Rights Reserved

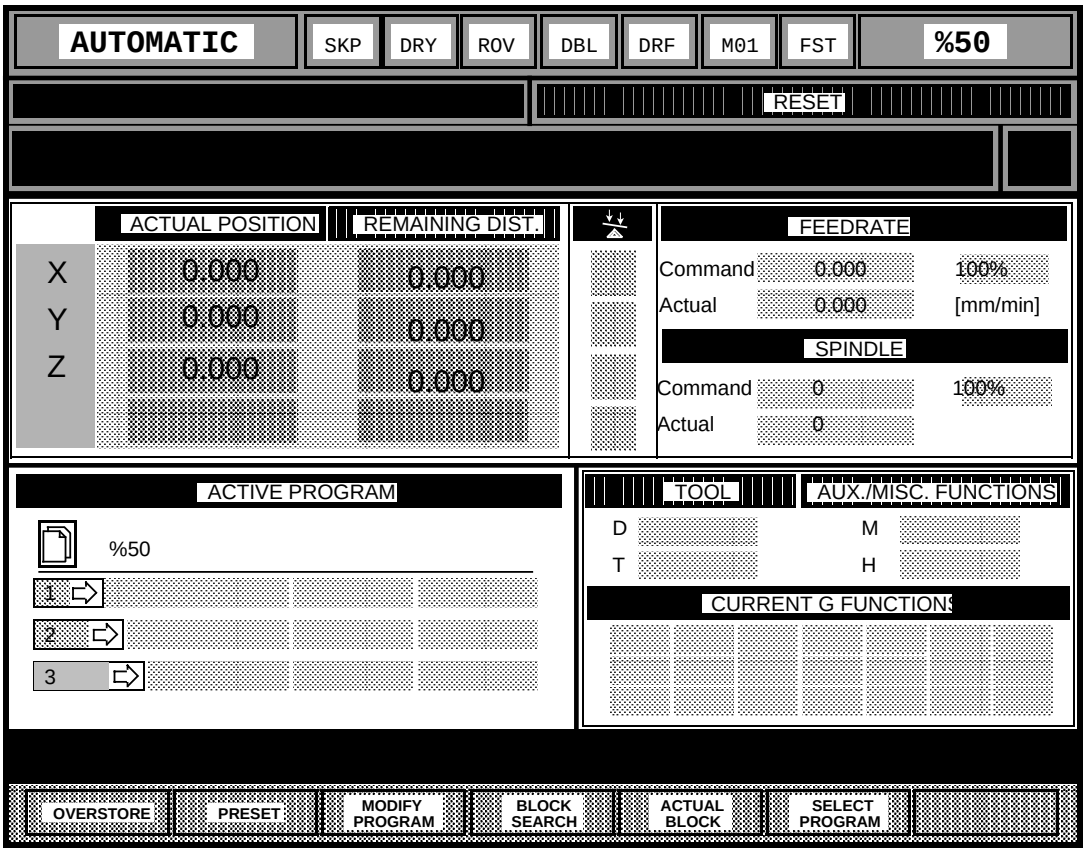
3.7 AUTOMATIC mode

The previously selected part program is executed in this mode. A part program (e.g. on punched tape) entered via the universal interface or directly through the operator keyboard. Other programs can be edited whilst a part program is being executed.

3.7.1 AUTOMATIC basic display

After AUTOMATIC mode has been selected using the following key:

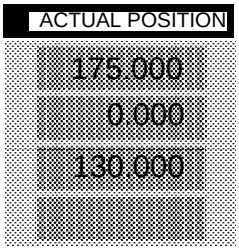
the following basic display appears:



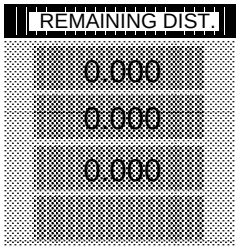
©Siemens AG 1990 All Rights Reserved 6ZB5 410-0EE02

3.7.2 Notes on AUTOMATIC basic display

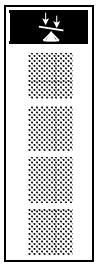
Display of the addresses of each of the axes. After the reference point of an axis has been set, the appropriate field is darkened.



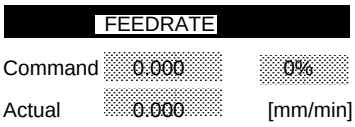
The actual position of each of the axes is shown in these fields.



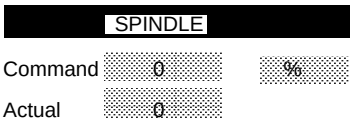
The distance to go arising during the processing of the commenced NC block, is displayed here.



Movement of an axis in a positive or negative direction is indicated by a plus or minus sign in the appropriate field.

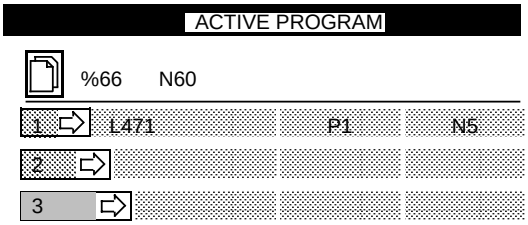


Display of the command and actual value of the feed rate and feed rate type as well as the position of the feedrate/rapid traverse override switch.



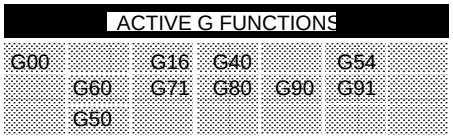
Display of spindle speed set and actual value as well as position of spindle sped override switch.

©Siemens AG 1990 All Rights Reserved



The number of the block now to be processed is displayed in this window.
In this example, the block to be processed is block N5 from subroutine L471.
Subprogram L471 has been called up from the main program %66 in block N60.

The active tool number (T) and the override number (D) as well as the M and H functions last programmed are displayed here.



This field displays the G functions which are currently valid or programmed.

Short description of softkeys:

OVERSTORE	:	Entry of auxiliary functions (s. Section 5.3.12)
PRESET	:	Set actual value (s. Section 5.3.13)
PROGRAM CONTROL	:	Modify sequence of a part program
BLOCK SEARCH	:	Program begin at the relevant point of the part program (s. Section 5.3.4)
ACTIVE BLOCK	:	Display of the active block contents (s. Section 5.3.2)
SELECT PROGRAM	:	Selection of part program for execution (s. Section 5.3.1)
OVERRIDE BLOCK	:	Display of part program block with possibility for editing (s. Section 5.3.14)

©Siemens AG 1990 All Rights Reserved 6ZB5 410-0EE02

3.8 TEACH IN mode

For software version 3 the TEACH IN mode is not yet available.

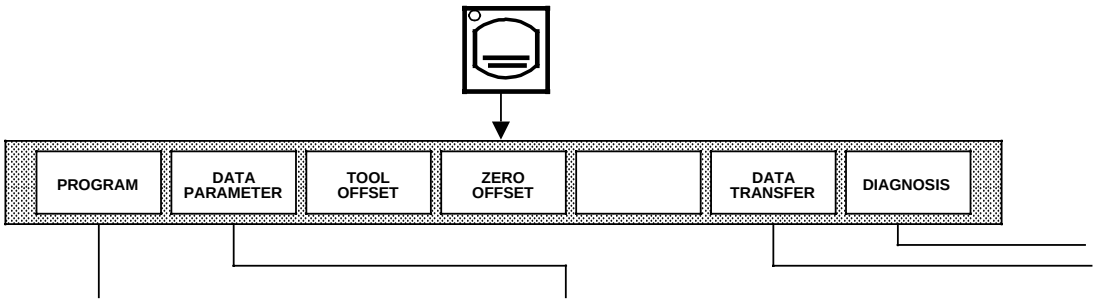
©Siemens AG 1990 All Rights Reserved

4 Data Input

4.1 General comments

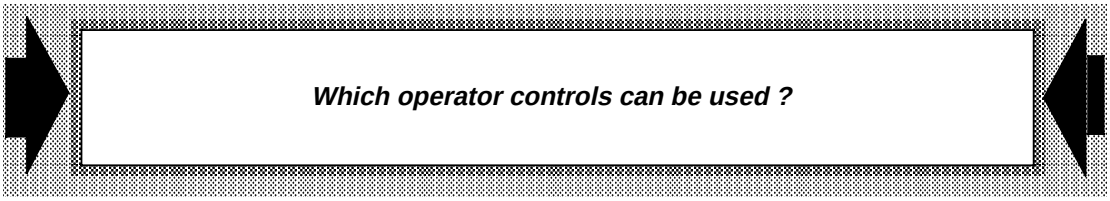
In the previous sections the functions of operator controls (Section 4) and operating modes (Section 3) are described.

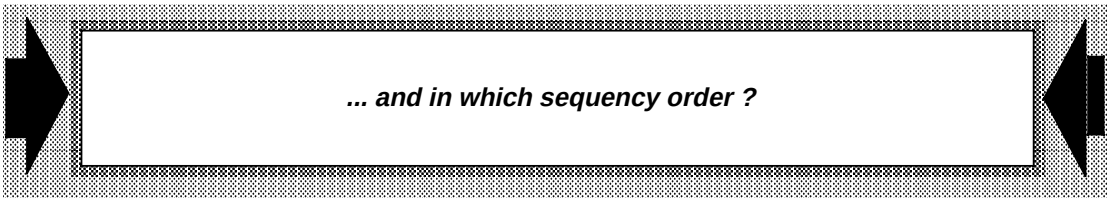
Section 4 explains the input of data for programs, parameters, tool offsets, zero offsets, data transfers and diagnostics.
Section 4 structure follows the menu tree structure ”data area”:



Section 4 and 5 inform about SINUMERIC 805 operation.

They explain:

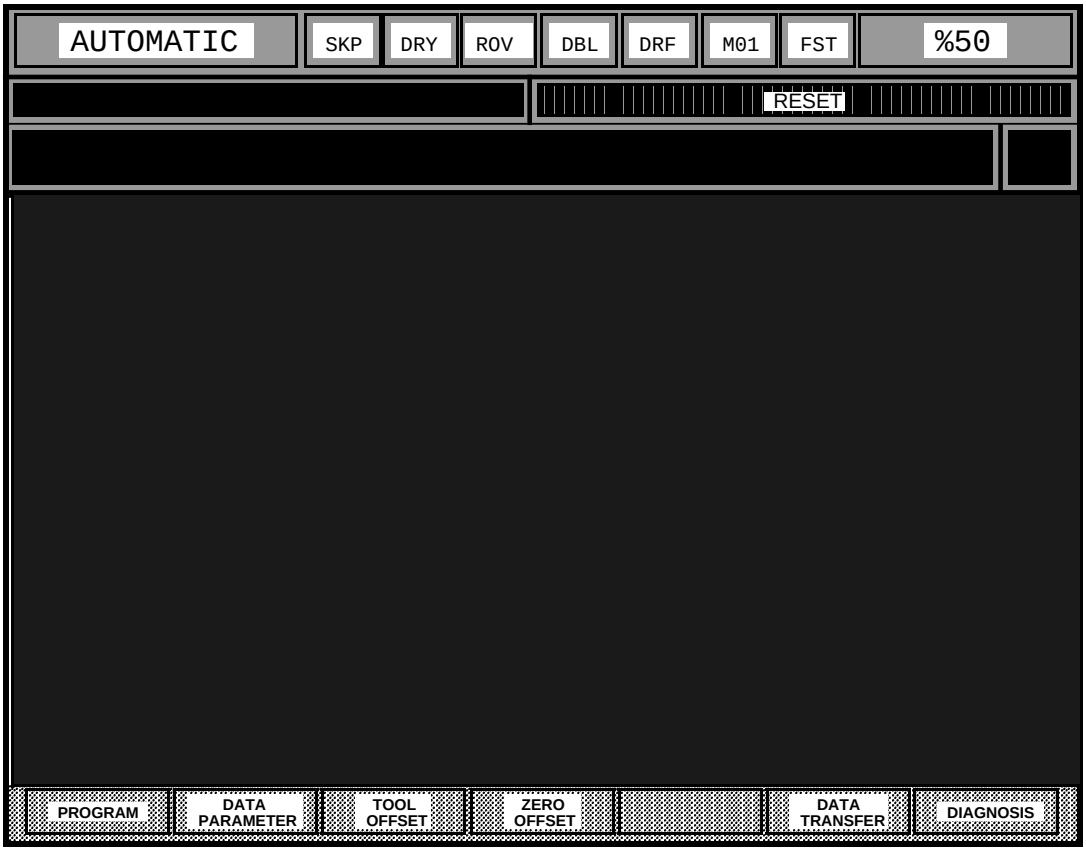




©Siemens AG 1990 All Rights Reserved

4.2 Data area basic display

The following display appears when the data storage area is selected using the operating area key on the SINUMERIK 805.



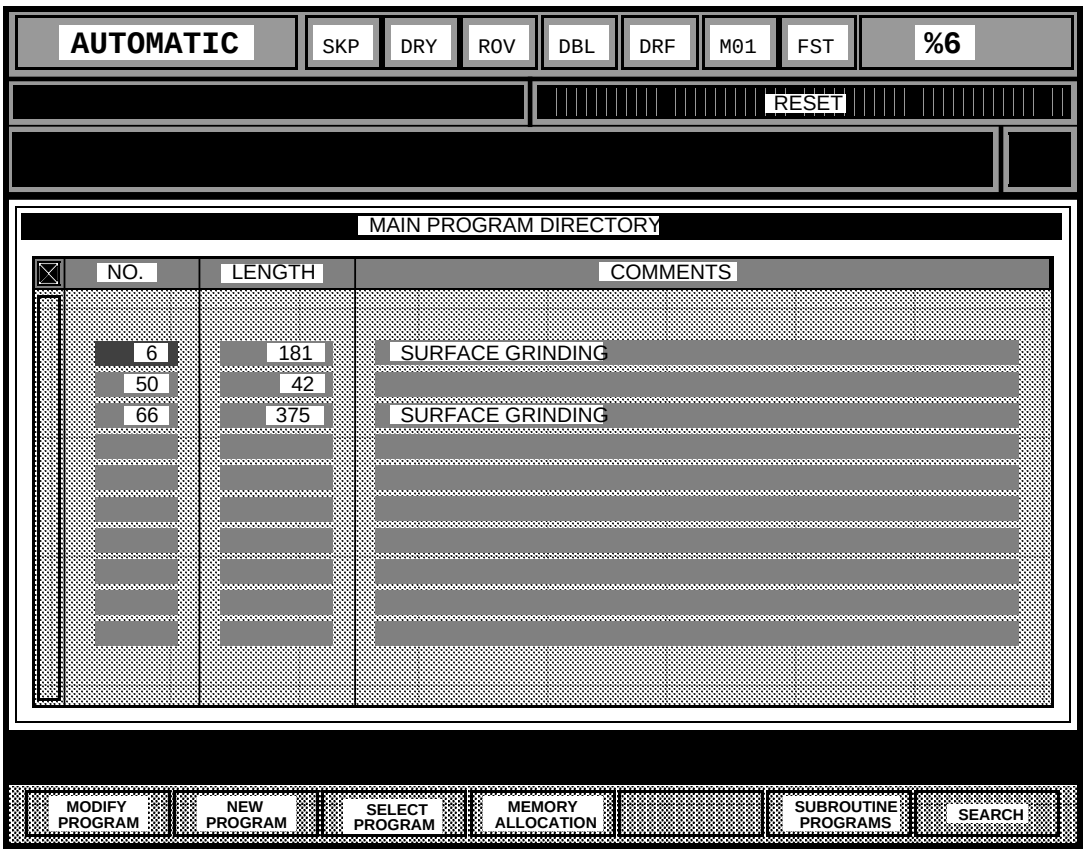
Notes:

- The key LED on the data area key lights up when the data area has been selected.
- The menu tree is shown in Section 2.4.2.
- See also Section 2.4 (Operating areas, menu tree).

©Siemens AG 1990 All Rights Reserved

4.3 Program

When the "PROGRAM" softkey in the data storage area basic display is operated, the following display appears:



Short description of the softkey bar:

MODIFY PROGRAM	Edit the program selected by cursor. (Section 4.3.1)
NEW PROGRAM	Create a new part program. (Section 4.3.2)
SELECT PROGRAM	When this softkey is operated, the program selected by means of cursor is selected for processing by the control. (Section 4.3.3)
MEMORY ALLOCATION	Indicates the user memory size (part program memory), the free and allocated user memory space. (Section 4.3.4)
SUBROUTINE PROGRAMS	Switchover to the MAIN PROGRAM DIRECTORY. (Section 4.3.5)
SEARCH	Direct input of the program number of the part program being searched for. (Section 4.3.6)

©Siemens AG 1990 All Rights Reserved

Softkeys in the menu extension bar:

COPY	:	Copy the selected program onto a new program number (Section 4.3.7).
DELETE	:	The program selected by cursor or by entering the program number can be deleted (Section 4.3.8).
RENAME	:	The selected program is given another number (Section 4.3.9).
RELOCATE	:	The selected program is pushed to the end of the memory for editing (Section 4.2.10)
REORGANIZE	:	The part program memory is reorganized (Section 4.3.11).

4.3.1 Program editing

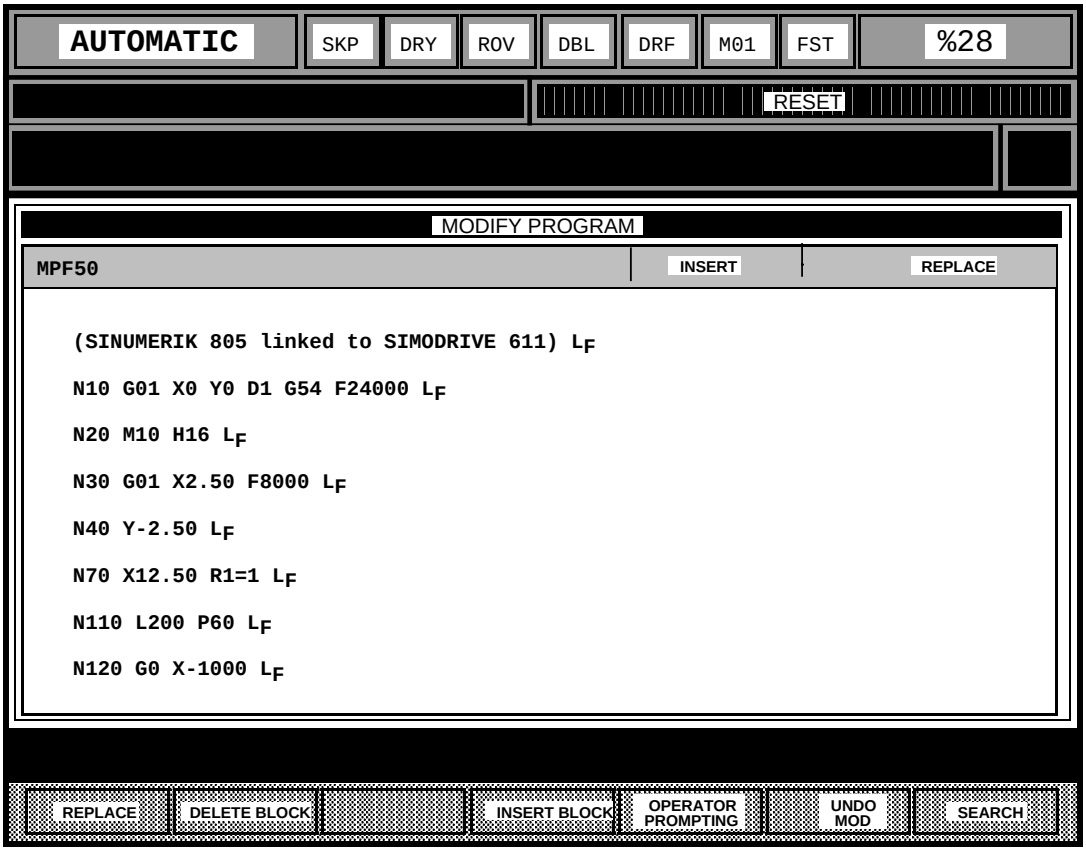
Sequence of operation

Prerequisite:: The desired program is selected in the OVERVIEW MAIN PROGRAMS/SUBROUTINES (s. Section 4.2.3).

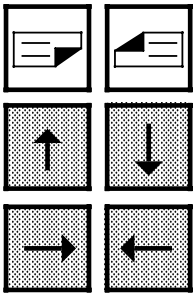
EDIT PROGRAM

Operate the PROGRAM EDIT key. If %50 is selected, for example, the following display appears:

Note:
Simultaneous to a running program (in this example %28) you can edit another program (in this example %50).



©Siemens AG 1990 All Rights Reserved



Use the page or cursor keys to move the cursor to the part of the part program which is to be modified.

You can delete the character selected by the cursor with the DELETE key.

INSERT BLOCK

Insert a new block after the block selected by the cursor.

DELETE BLOCK

Delete the block selected by the cursor.

REPLACE

Alternate between insert and replace mode.

Insert mode : The keyed in characters are inserted in the part program in front of the point where the cursor is positioned.

Overwrite mode : The keyed in characters overwrite those already contained in the part program from the point where the cursor is positioned.

UNDO MOD

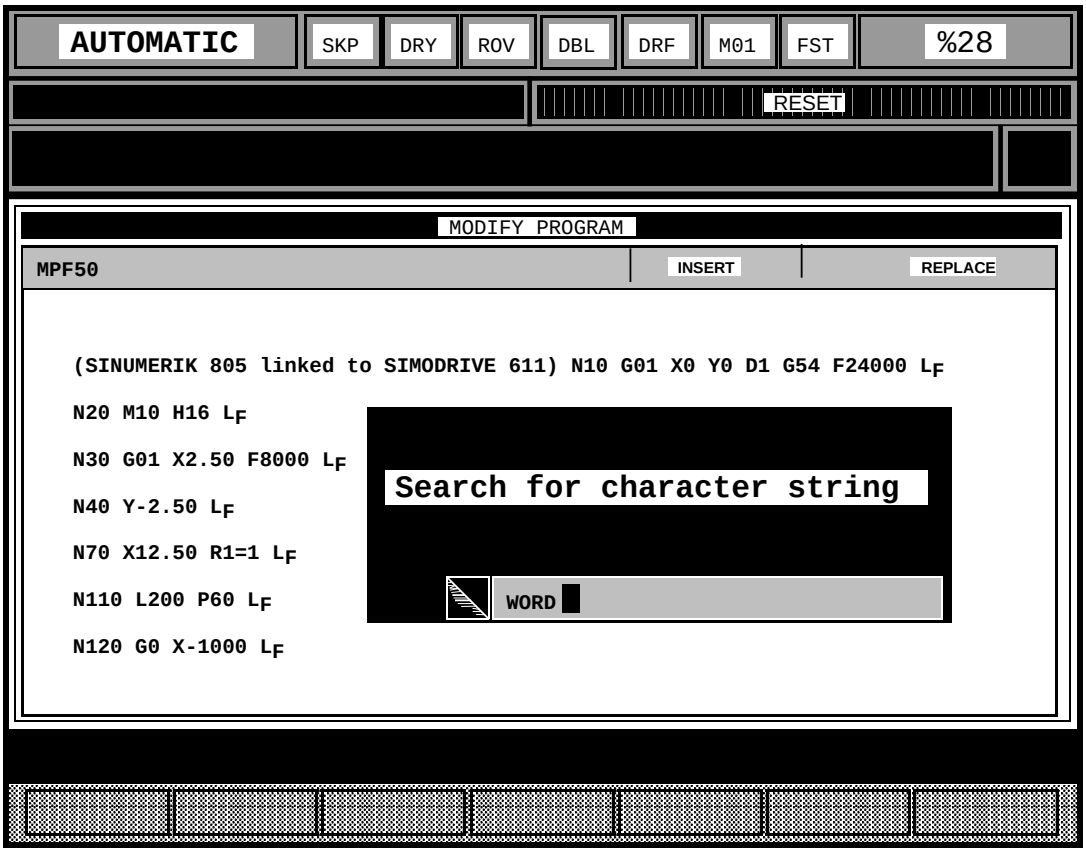
The most recent modifications in the text are reversed. This is only possible if the INPUT key has not been operated.

Use the INPUT key to transfer the modification of the block into memory.

©Siemens AG 1990 All Rights Reserved

SEARCH

When the SEARCH key is operated, the following window appears:



The character string being searched is keyed in via the keyboard.

When the INPUT key is operated, the entry is entered into the control and the cursor positioned on the character string being searched for.

©Siemens AG 1990 All Rights Reserved

4.3.1.1 OPERATOR PROMPTING in edit mode

The OPERATOR PROMPTING function is to activate via NC-MD 5150.2.

This function is a user input support for parameter assignment at Siemens Standard Cycles and User Cycles.
For each subroutine programmable with R parameter the user can install OPERATOR PROMPTING function (see Programming Guide, Section 12). The OPERATOR PROMPTING can be called in MDA and in edit mode.

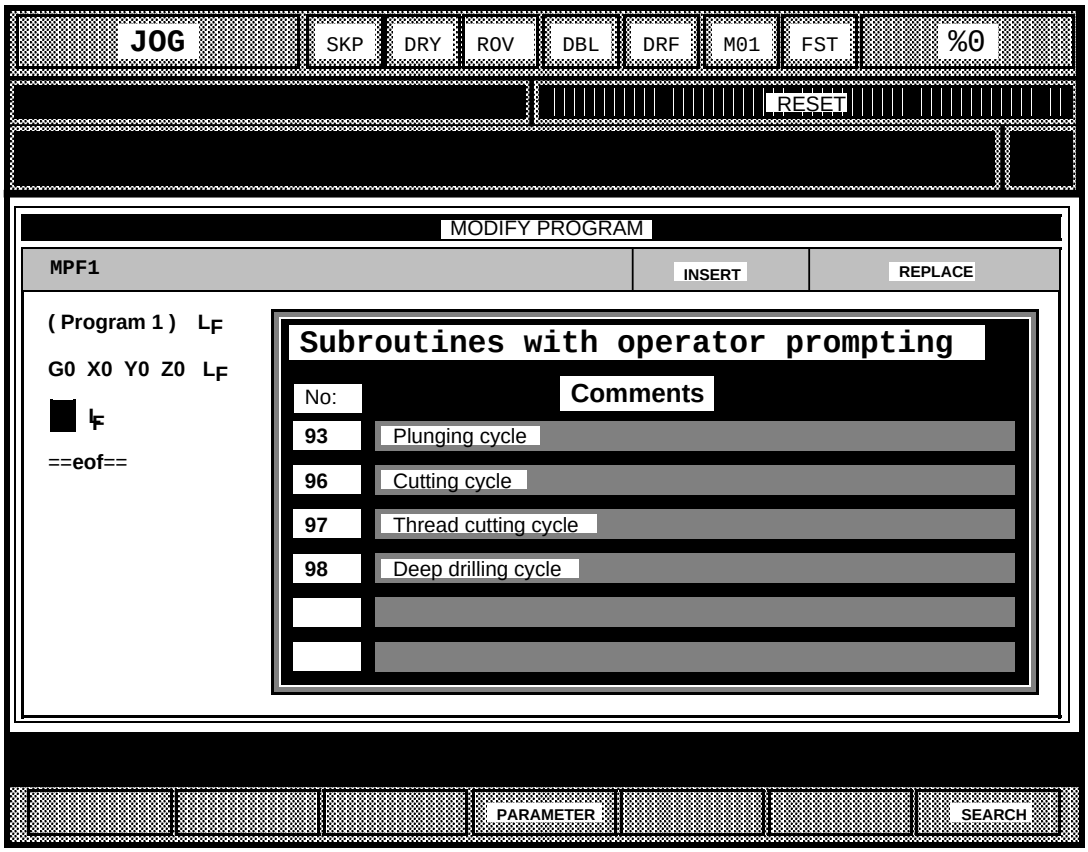
a) Manual subroutine selection

Sequence of operation

Prerequisite:: Edit mode has been selected (Section 4.3.1 resp. 4.3.2)

OPERATOR PROMPTING

Operation of OPERATOR PROMPTING softkey opens a window listing all subroutines fitted out with operator prompting.



©Siemens AG 1990 All Rights Reserved


Exist more than 6 subroutines with operator prompting they can be displayed by operating PAGE UP/DOWN softkeys.


Using the cursor keys you can select the wanted subroutine in the overview display (inverse displayed ≙ selected).

.

b) Subroutine selection using the SEARCH softkey

Sequence of operation

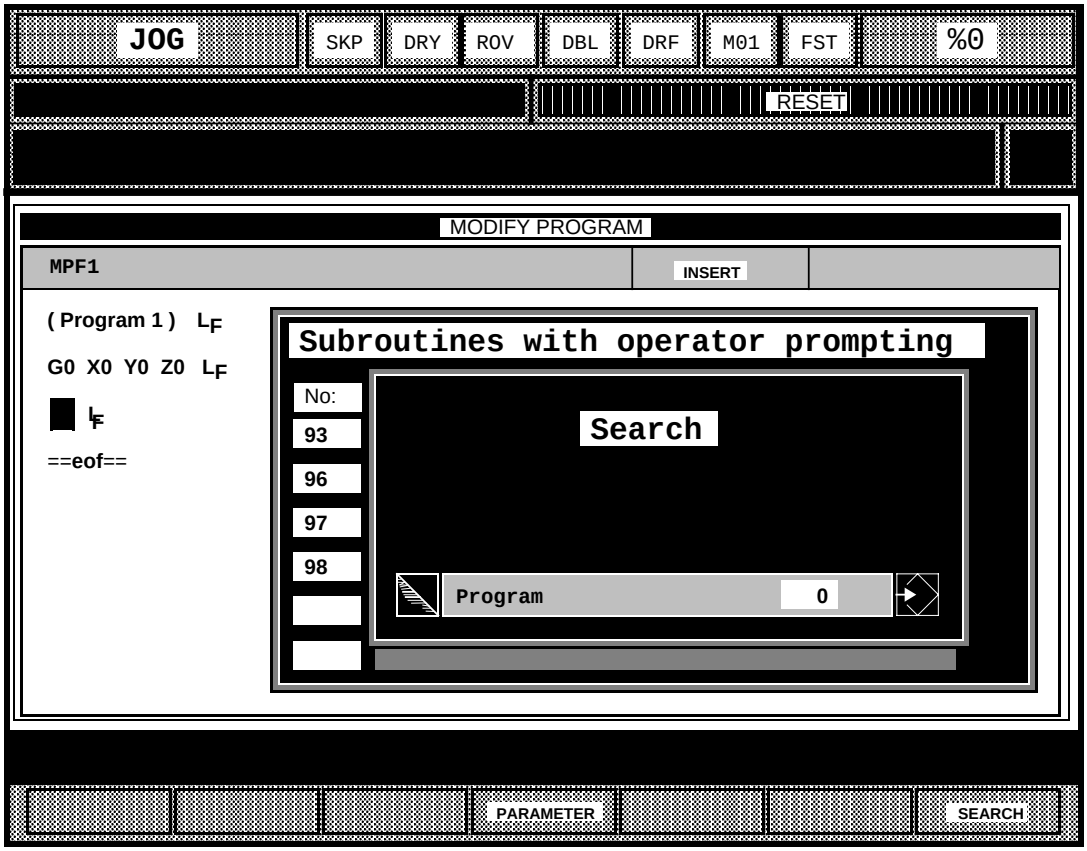
Prerequisite: MDA mode has been selected.

OPERATOR PROMPTING

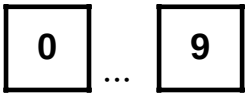
Operation of OPERATOR PROMPTING softkey opens a window listing all subroutines fitted out with operator prompting.

SEARCH

Operate the SEARCH softkey. The following window appears:



©Siemens AG 1990 All Rights Reserved



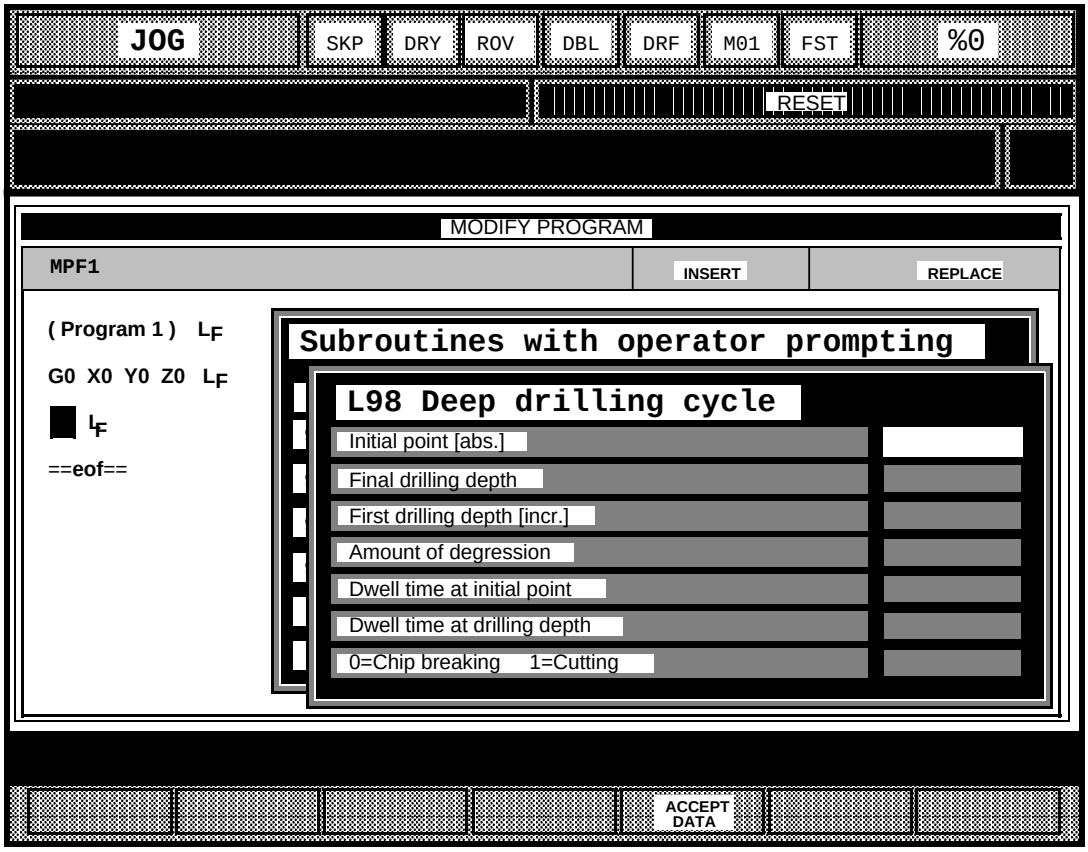
Key in the desired subroutine number in the opened entry field.

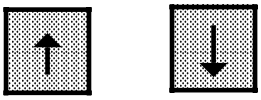
Operation of INPUT key closes entry field and window.
Now the wanted subroutine is selected (inverse displayed).

Parameter assignment for the selected subroutine

Prerequisite: The wanted subroutine has been selected.

Operation of PARAMETER softkey opens another window listing all subroutines fitted out with operator prompting.





Use cursor keys for R parameter selection.

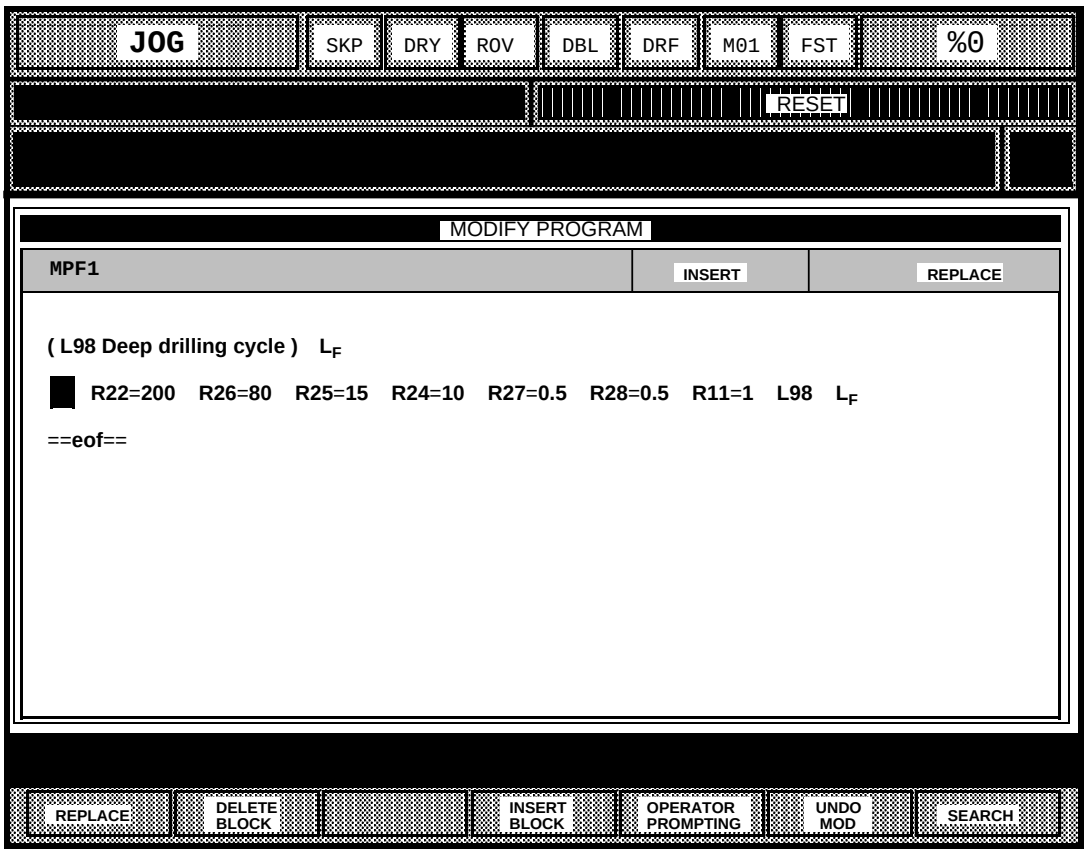
©Siemens AG 1990 All Rights Reserved

Now entry the parameters after description in Section 2.4.4 (Entries in Input fields).

Exist more than 7 parameters for the subroutine they can be displayed by operating PAGE UP/DOWN softkeys.

ACCEPT DATA

After operating the ACCEPT DATA softkey the part program accepts the entried R parameter assignments.
After all R parameter assignments the subroutine call automatically is entried.
The PARAMETER and DIRECTORY windows are closed.
You are back in edit mode.



Note:

After operation of REPLACE softkey the error message ”STOP NOT ALL PARAMETERS ENTRIED” appears when not all parameters are entried.
The window is closed by operating key.

©Siemens AG 1990 All Rights Reserved

4.3.2 Create new program

Sequence of operation

Prerequisite: The MAIN PROGRAM/SUBROUTINE DIRECTORY has been selected (see Section 4.3.5).

NEW PROGRAM

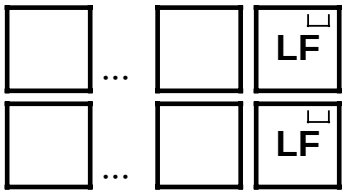
The following window (in the case of main programs) appears when you operate the NEW PROGRAM softkey:

0 ... 9

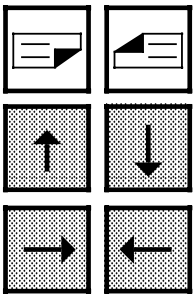
Now key in the program number of the part program to be created.

Operating the INPUT key closes the input field and takes you into the EDIT mode.

©Siemens AG 1990 All Rights Reserved



You can now enter one block after the other through the keyboard. You must, however, complete every block by pressing the ... LF key.



Use the page or cursor keys to move the cursor to the part of the part program which is to be modified.

You can delete the character selected by the cursor with the DELETE key.

INSERT BLOCK

Insert a new block after the block selected by the cursor.

DELETE BLOCK

Delete the block selected by the cursor.

REPLACE

Alternate between insert and overwrite mode.

Insert mode	:	The keyed in characters are inserted in the part program from the point where the cursor is positioned.
Overwrite mode	:	The keyed in characters overwrite those already contained in the part program from the point where the cursor is positioned.

OPERATOR PROMPTING

Programming support for subroutines (see Section 4.3.1.1).

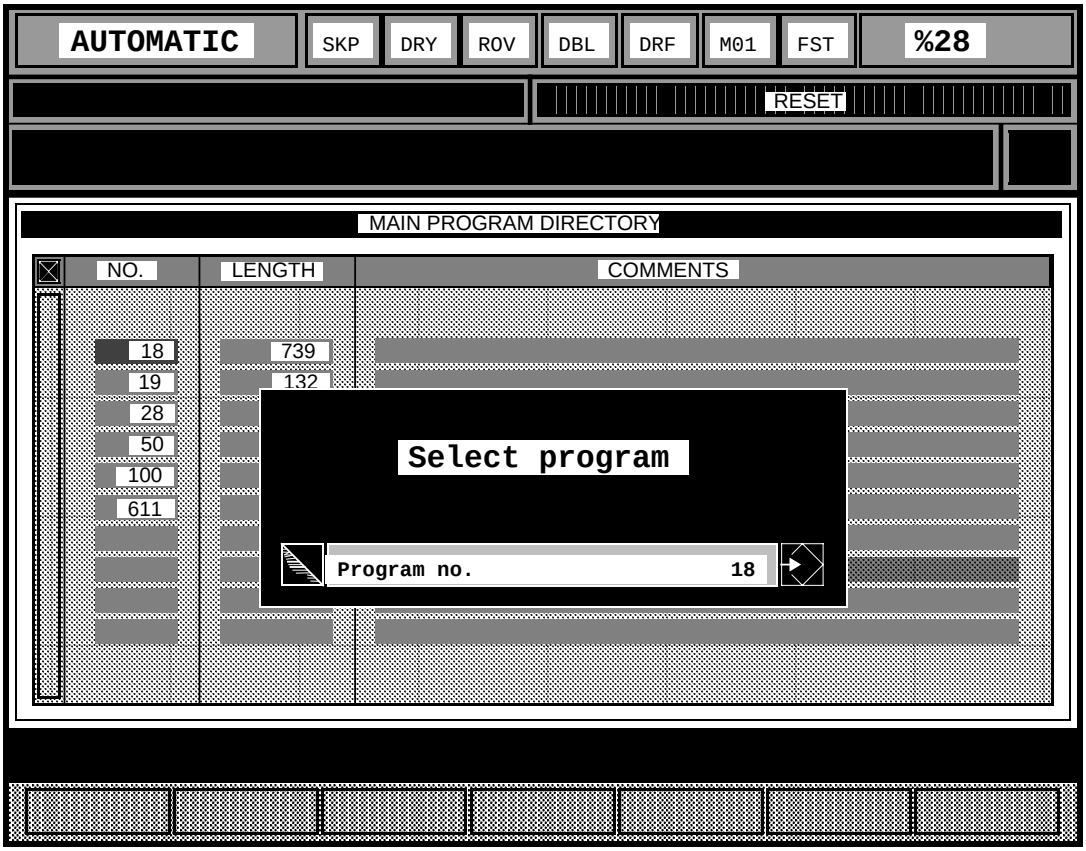
©Siemens AG 1990 All Rights Reserved

4.3.3 Part program selection for program execution

You can use the SELECT PROGRAM softkey to select a specific main program contained in the program memory for program execution in the AUTOMATIC mode.

Sequence of operation

SELECT PROGRAM	Press the SELECT softkey. The program number of the program selected in MAIN PROGRAMS OVERVIEW now appears in the field designated for program numbers (s. Section 2.3).



0 ... 9	Now key in the program number in the opened input field.
	Operating the INPUT key closes the input field and takes you into the EDIT mode.

Note:

You can also select a program for program execution in the basic menu of the AUTOMATIC mode (s. Section 5.3.1).

©Siemens AG 1990 All Rights Reserved 6ZB5 410-0EE02

4.3.4 Memory allocation of the part program memory

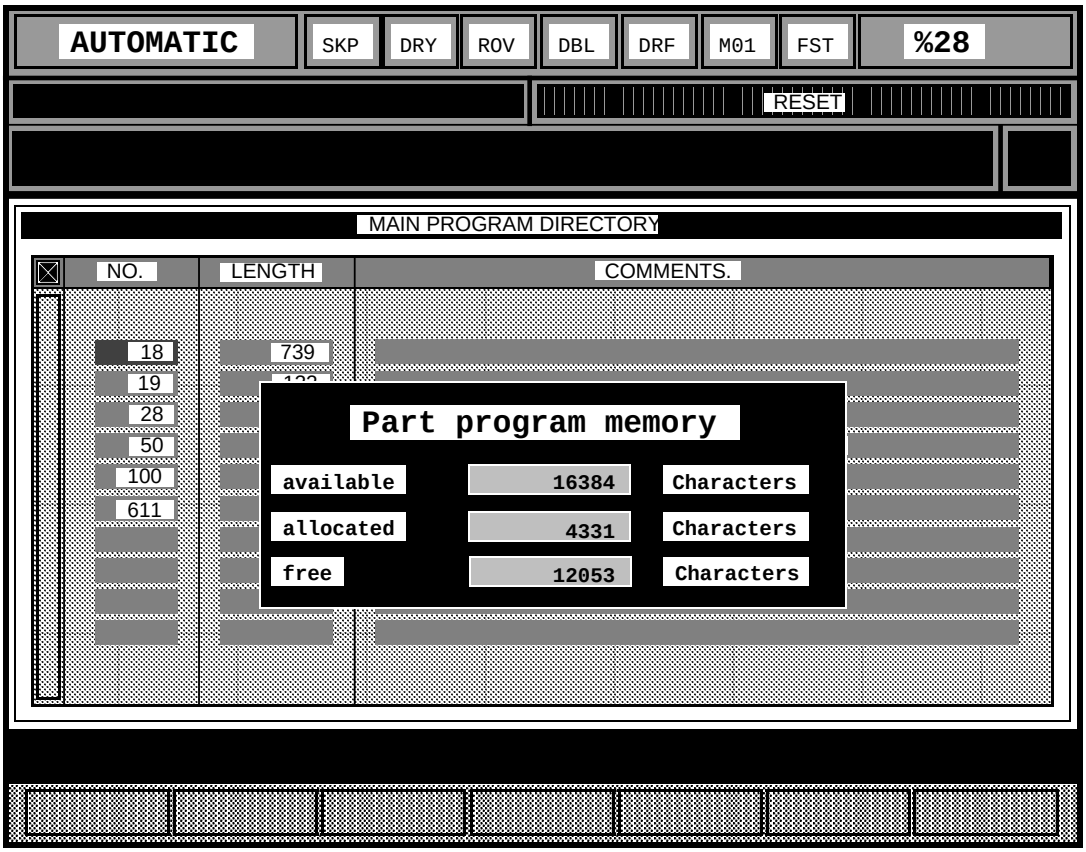
By operating the MEMORY ALLOCATION softkey in the main program or subroutine directory, you can call up information regarding the memory allocation of the program memory.

Sequence of operation:

Prerequisite: The MAIN PROGRAM or SUBROUTINE DIRECTORY is selected (s. Section 4.2.5).

MEMORY ALLOCATION

You open the following window when you press the MEMORY ALLOCATION softkey.



You close the window by pressing the RECALL button. The overview previously selected appears on the screen.

Note:

After selecting the DATA TRANSFER basic menu (s. Section 4.6), the available, allocated and free memory space in the part program memory is displayed as above.

©Siemens AG 1990 All Rights Reserved

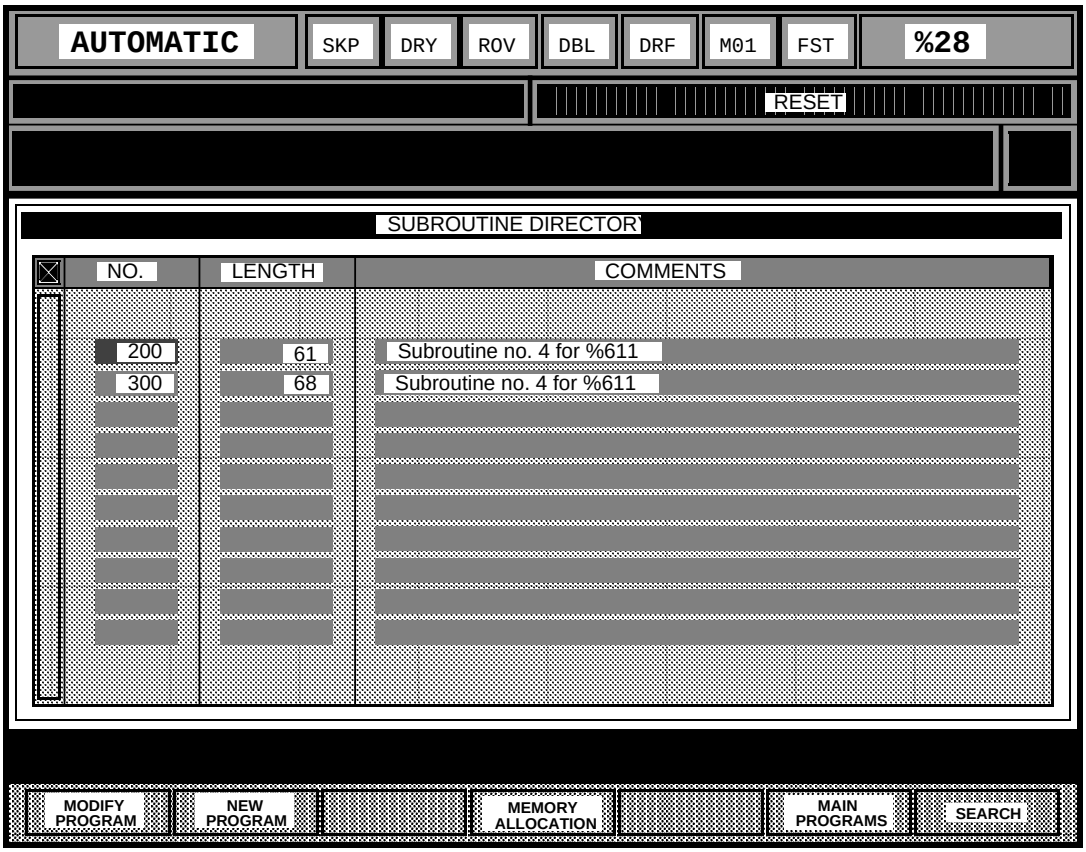
4.3.5 Display of MAIN PROGRAM DIRECTORY or SUBROUTINE DIRECTORY

An overview of all stored main programs is displayed when the basic menu PROGRAM is selected. Using a softkey, you can alternate between MAIN PROGRAM DIRECTORY and SUBROUTINE DIRECTORY as follows:

Sequence of operation:

SUBROUTINE PROGRAMS

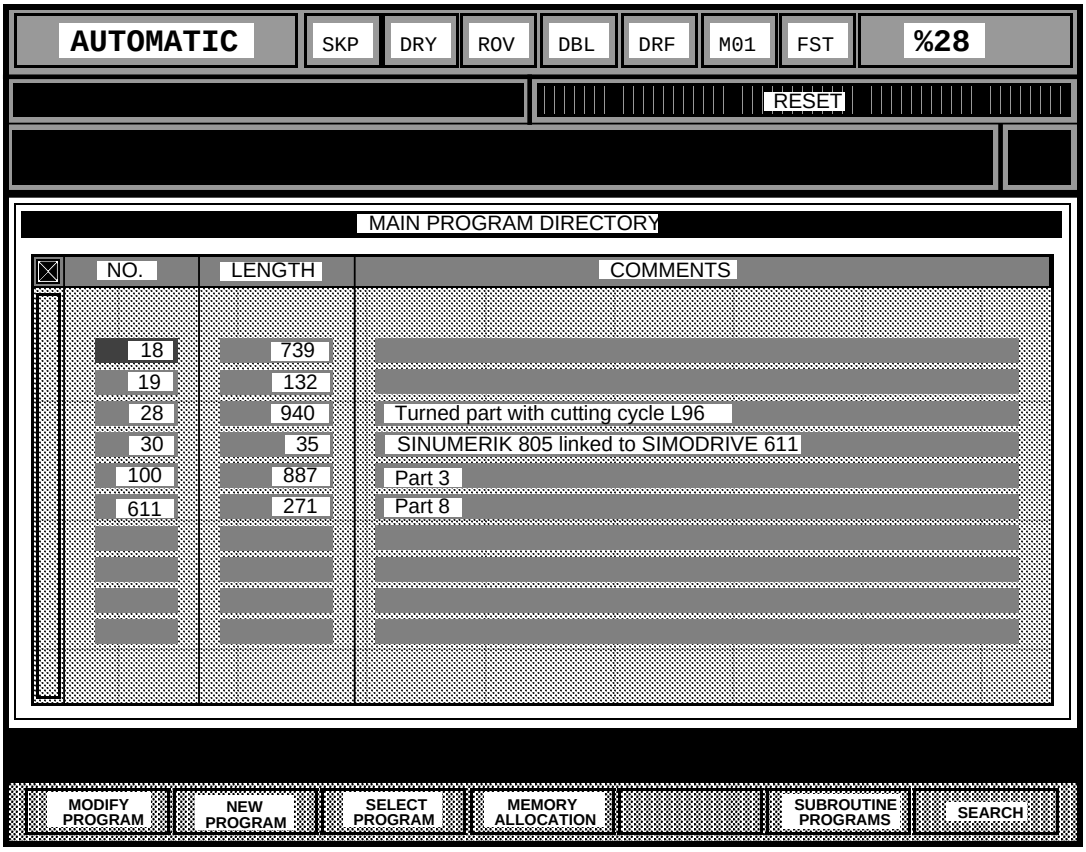
MAIN PROGRAM DIRECTORY has been selected. The display changes accordingly when the softkey SUBROUTINE DIRECTORY is operated:



©Siemens AG 1990 All Rights Reserved

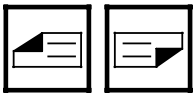
MAIN PROGRAMS

SUBROUTINE DIRECTORY has been selected. The display changes accordingly when the softkey MAIN PROGRAMS has been selected:



Notes:

- A maximum of 10 main programs and/or subroutines together with program no., length and comment can be displayed in the first screen display.

- 

  When the screen display is full, additional overviews of available main programs and subroutines can be displayed by operating the PAGE UP/DOWN keys.

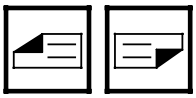
©Siemens AG 1990 All Rights Reserved

4.3.6 Selecting a program in the overview

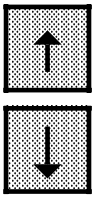
If you wish to select, display or modify a main program or display or modify a subroutine, you can select the required program in the MAIN PROGRAM or SUBROUTINE DIRECTORY manually or by means of the SEARCH softkey as follows:

a) Manual program selection

Sequence of operation when selecting a program manually:



Use the PAGE UP/DOWN keys to select the directory display which contains the required main program or subroutine.



Use the cursor keys to select the desired main program or subroutine within one directory display (displayed in reverse video = selected).

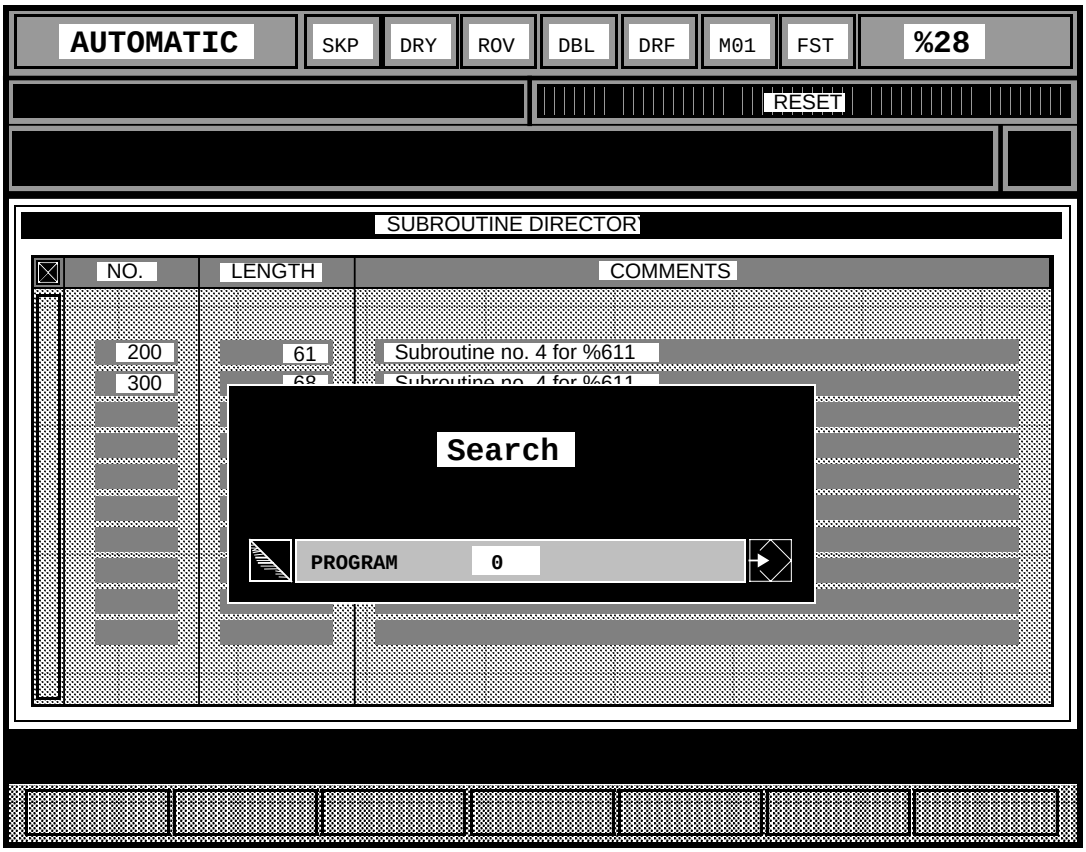
©Siemens AG 1990 All Rights Reserved

b) Program selection using SEARCH softkey

Sequence of operation

SEARCH

Operate the SEARCH softkey to open the following window:



0 ... 9

Now key in the number of the desired main program or subroutine in the opened input field using the numeric keys.

Use the INPUT key to close the input field and the window. The required program is now selected in the MAIN PROGRAM DIRECTORY or in the SUBROUTINE DIRECTORY (displayed in reverse video).

©Siemens AG 1990 All Rights Reserved

4.3.7 Copy part programs

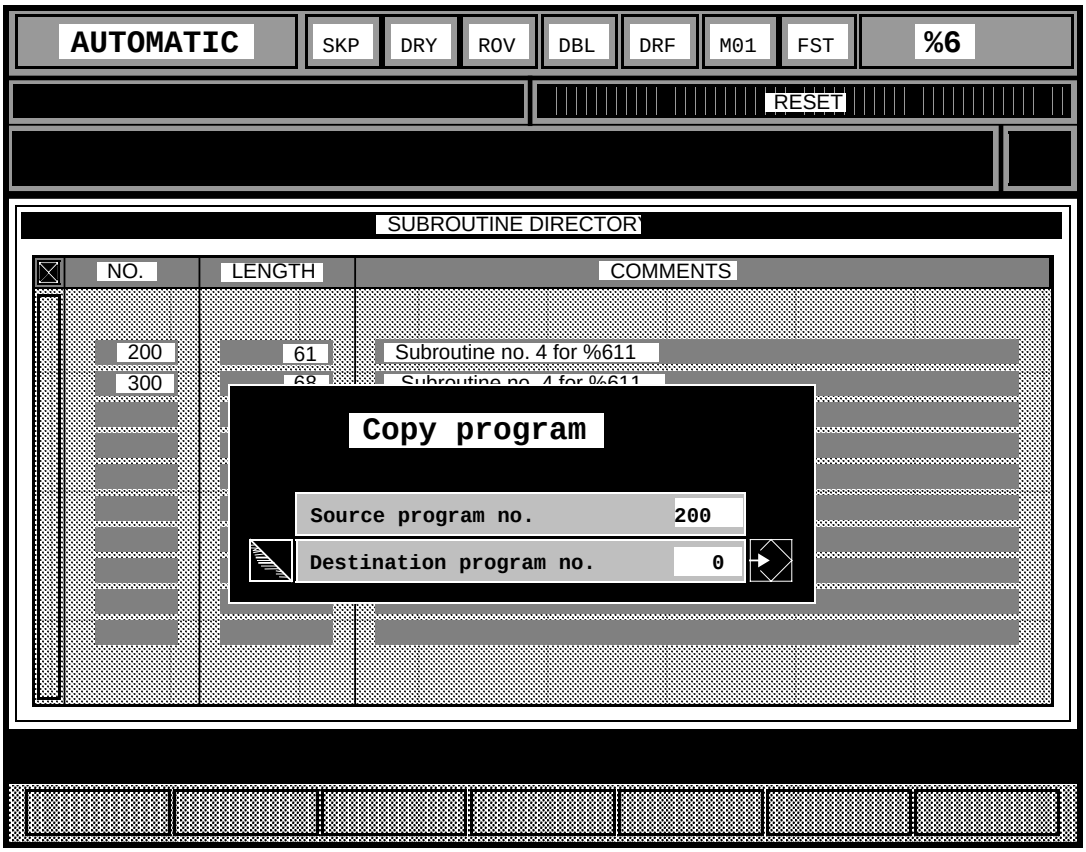
You can additionally store a part program under a different program number using the COPY softkey.

Sequence of operation

Prerequisite: The required program is selected in the MAIN PROGRAM or SUBROUTINE DIRECTORY (Section 4.3.6)

COPY

By operating the COPY softkey, you open the following window:



0 ... 9

Now key in the program number by which the copied program is to be known.

Operate the INPUT key to end the entry and execute the copy function.

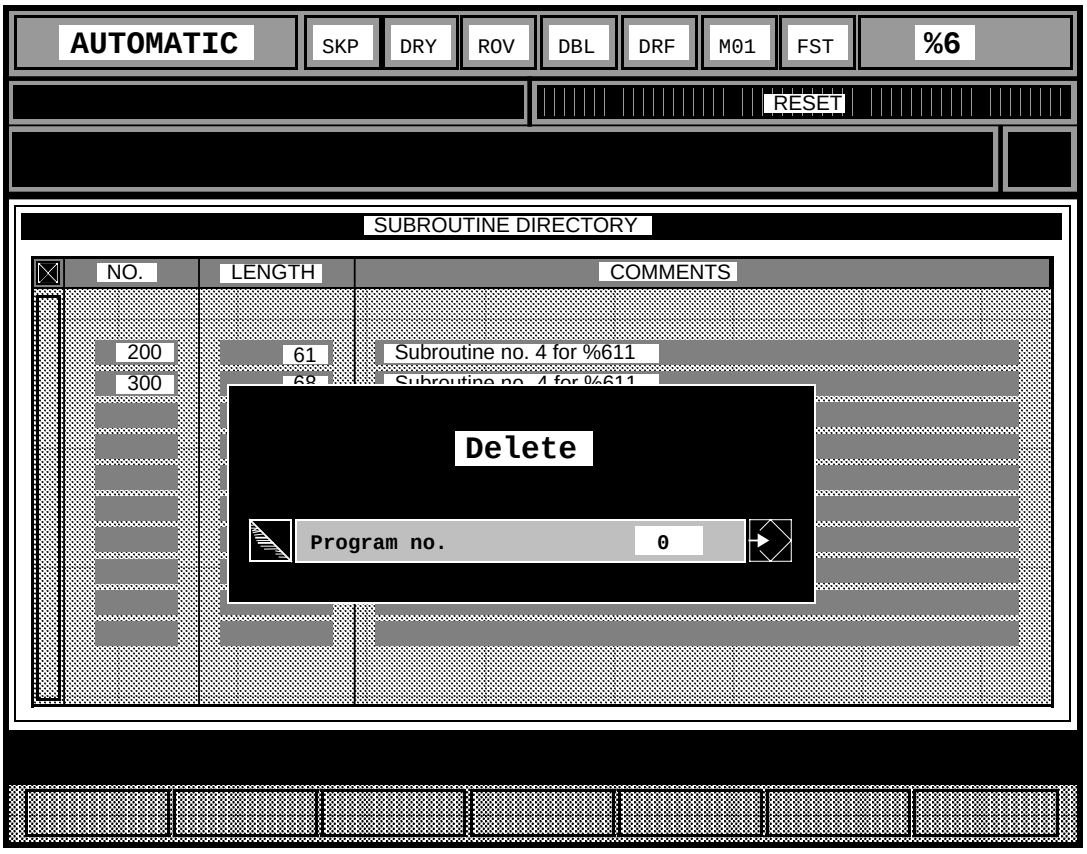
4.3.8 Delete part program

Sequence of operation

Prerequisite: The MAIN PROGRAM or SUBROUTINE DIRECTORY is selected.

DELETE

By operating the DELETE softkey, you open the following window:



0 ... 9

Now key in the program number of the part program to be deleted.

Operate the INPUT key to end the entry and execute the delete function.

©Siemens AG 1990 All Rights Reserved

4.3.9 Rename part program

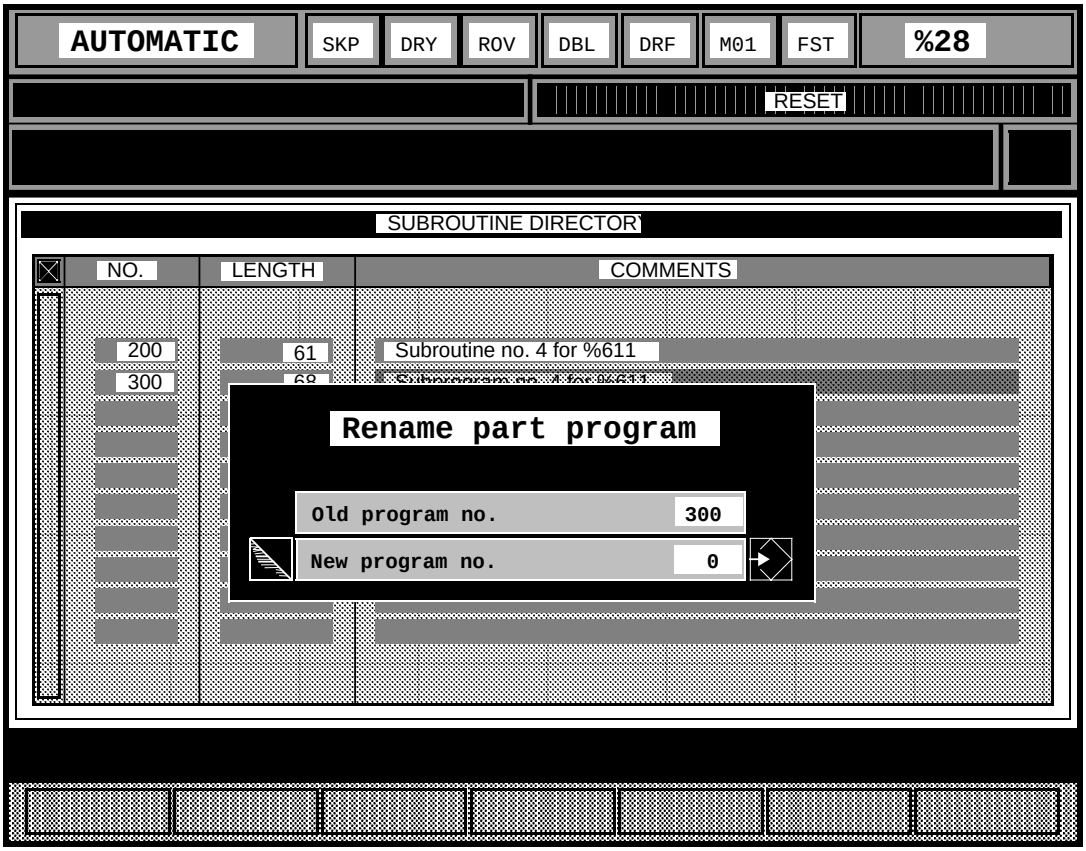
You can use the RENAME function to change the program number of a part program.

Sequence of operation

Prerequisite: The required program has been selected in the MAIN PROGRAM OR SUBROUTINE DIRECTORY.

RENAME

Operate the RENAME key to open the following window:



0 ... 9

Now key in the new program number.

Operate the INPUT key to end the entry and execute the function.

©Siemens AG 1990 All Rights Reserved

4.3.10 Relocate part program

You use the **MOVE** function to shift the selected program to the end of the memory. This is necessary in the following cases:

1) You wish to edit the program whilst processing another part program. The program to be edited must be shifted with the **MOVE** function.

2) You wish to edit the program and extend it by 1500 characters.

Sequence of operation

MOVE	Operate the softkey MOVE. The function is executed without a message.

4.3.11 Reorganize part program memory

Any memory made available when part programs have been deleted is not taken into account when the memory is allocated. You can use the function **REORG** to reorganize the contents of a program memory. Thus, any memory space made available by deleting programs can be used again for program entries (see also alarm no. 31: No further part program input possible).

Sequence of operation

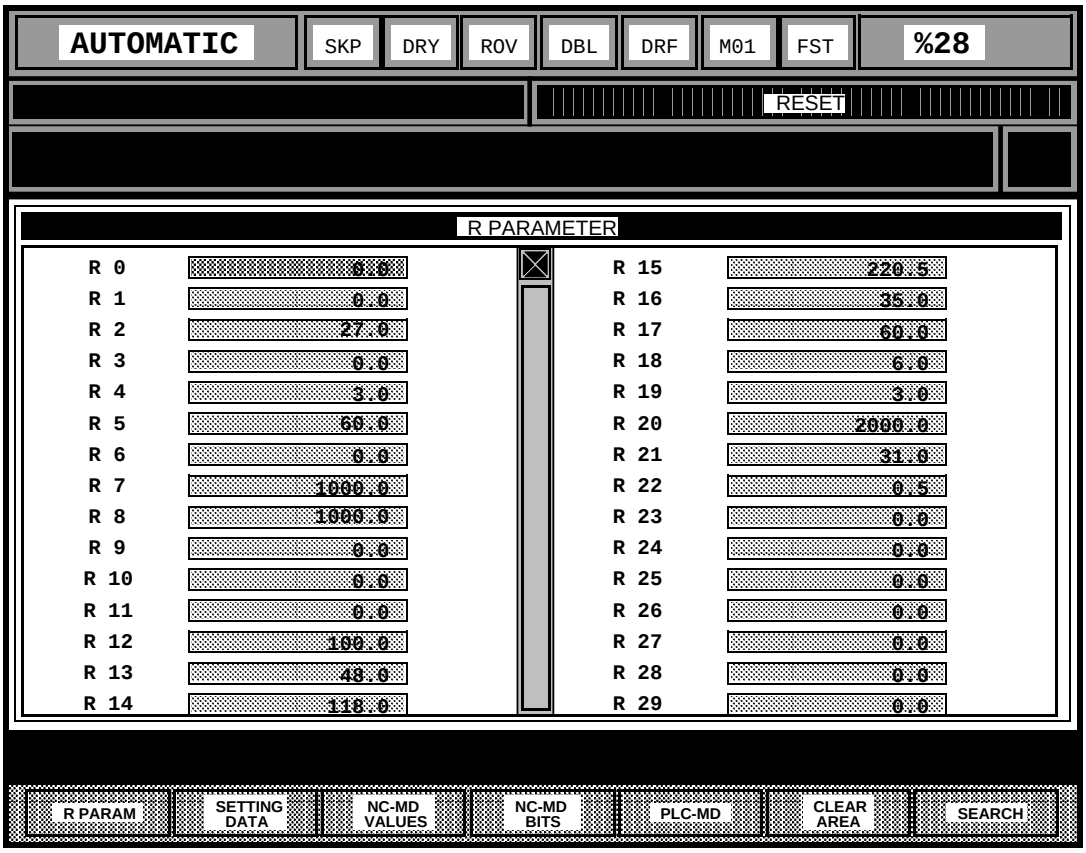
REORGANIZE	Operate the softkey REORGANIZE. The function is executed with the message "Part program is being reorganized.

The free memory space is checked using the MEMORY ALLOCATION function (Section 4.3.1).

©Siemens AG 1990 All Rights Reserved

4.4 Data/Parameters

The following display appears on the screen when the softkey DATA/PARAMETER is operated in the basic display of the data area:



Data/parameter basic display

Short discription of softkeys:

R PARAM :
SETTING DATA :
NC-MD VALUES :
NC-MD BITS :
PLC-MD :
} Operate the appropriate softkey to select the desired data type.

CLEAR AREA : Clear an area which can be input.
SEARCH : Selection of a particular data parameter.

Notes:

- The selected data type is displayed in reverse video.
- The R parameter data type is selected in the DATA/PARAMETER display.
- The moveable input field is dark.

©Siemens AG 1990 All Rights Reserved

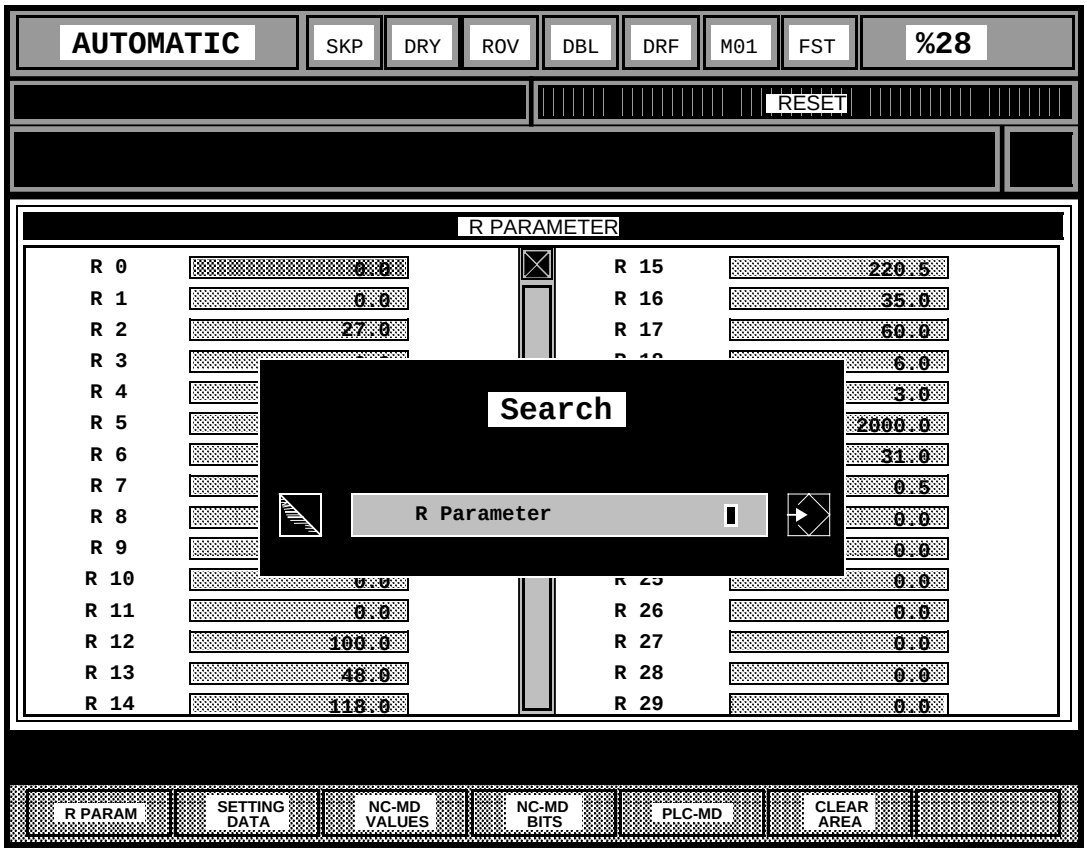
4.4.1 R parameters

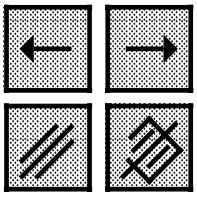
Sequence of operation for the selection of an R PARAMETER using the SEARCH softkey.

Prerequisite: The R PARAM data type is selected (either by selecting the data area or by operating the softkey R PARAM).

SEARCH

You open the following window by operating the SEARCH softkey:





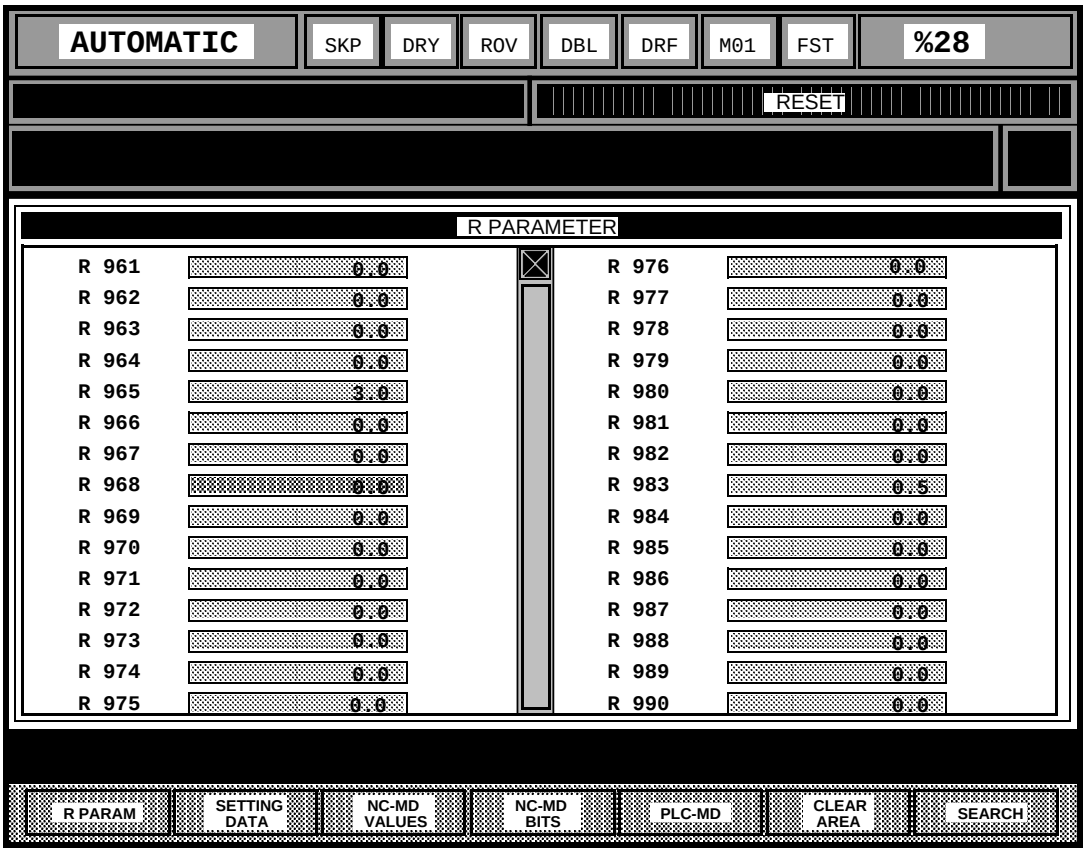
Now key in the number of the desired R PARAMETER in the input field which has already been opened using the numeric keys.

Use cursor and edit keys where necessary.

You enter the value and close the window using the INPUT key.

©Siemens AG 1990 All Rights Reserved

The following display appears when e.g. R968 is selected:



The value R968 appears in the darkened R968 input field. You can alter this value as follows.

Sequence of operation for altering an R PARAMETER value

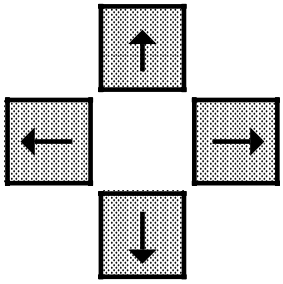
Prerequisite: The input field for the R PARAMETER to be changed has been selected using the SEARCH softkey or manually.

Notes:

- R PARAMETER values can be modified as described in section 2.4.4 (Entry in input fields).
- The calculator function (Section 2.4.5) can be applied to R PARAMETERS.

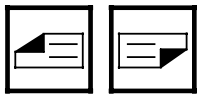
©Siemens AG 1990 All Rights Reserved

Sequence of operation when selecting an R PARAMETER manually



You move the dark input field between R PARAMETERS R0 and R999 using these keys.

You move the input field to the parameter situated above left by operating the HOME key.



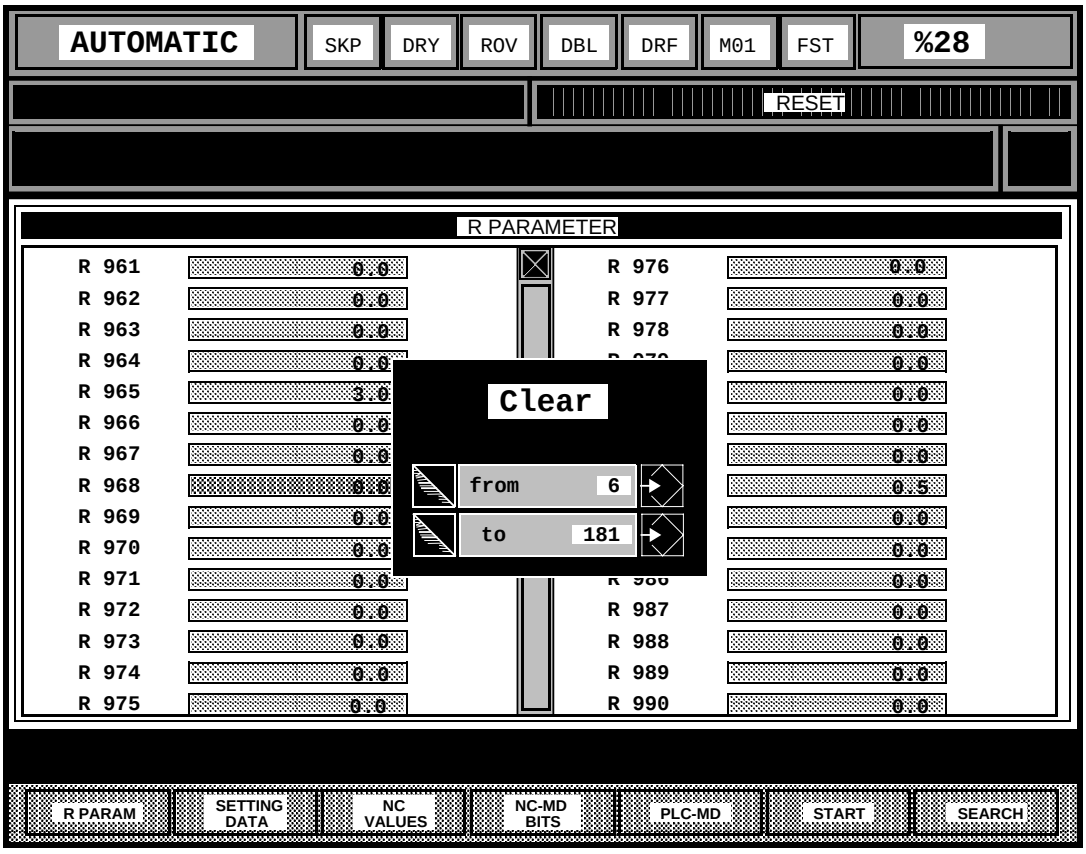
You page one display at a time forwards or backwards using these keys.

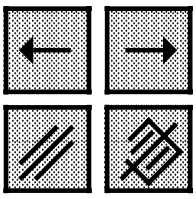
©Siemens AG 1990 All Rights Reserved

Sequence of operation when clearing an R PARAMETER area

CLEAR AREA

Operate the softkey CLEAR AREA to open the following window:



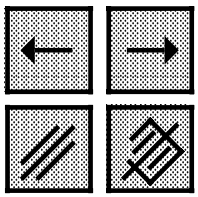


Now key in the first R PARAMETER to be deleted using the numeric keys in the opened field "from".

Use the cursor and edit keys where necessary.

Transfer the keyed in value using the INPUT key.

©Siemens AG 1990 All Rights Reserved



Key in the number of the last R PARAMETER to be deleted using the numeric keys in the opened field "to".

Use the cursor and edit keys where necessary.

Enter the keyed in value using the INPUT key.

START

Operate the softkey START to initiate the delete process of the input R parameter area and close the window. All R parameters of the deleted area now have the value 0.

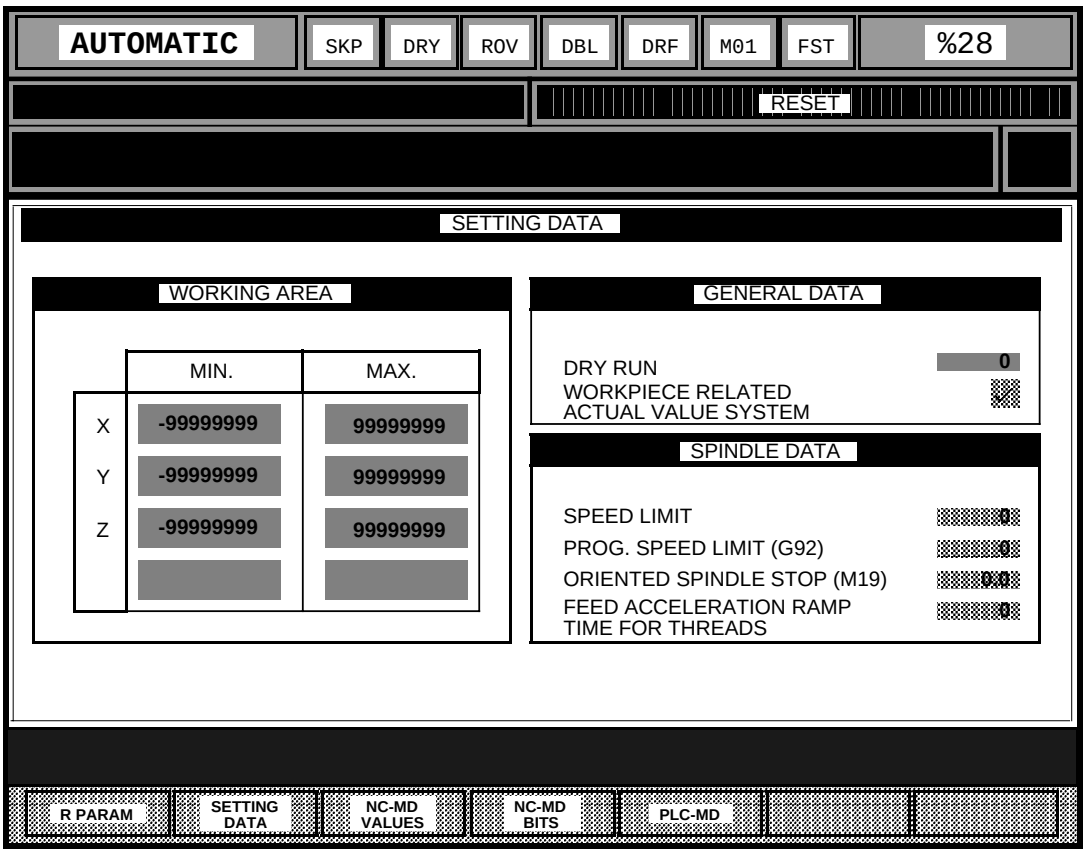
Note:

Before operating the softkey START, you can close the window using the RECALL  key. The delete process is not executed.

©Siemens AG 1990 All Rights Reserved

4.4.2 Setting data

When you have selected the data type SETTING DATA using the appropriate softkey, the following display appears:



Sequence of operation for altering setting data

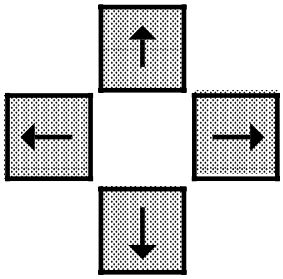
To modify or enter setting data, please follow the instructions in Section 2.4.4 (Entry in input fields).

© Siemens AG 1990 All Rights Reserved

Description	Standard value	Maximum input value	Reference system	Input unit
Dry run	0	24 000	IS	1000 units/min
Feed acceleration ramp time for threads	0	5		-
Min. working area limit	0	+/- 99 999 999	IS	units/min
Max. working area limit	0	+/- 99 999 999	IS	units/min
Prog. speed limit	0	12 000		rev/min
Oriented spindle stop	0	359.9		1/10 degrees
Spindle speed limit	0	12 000		rev/min

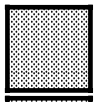
1) The "Maximum input value" is defined in MD!
2) The "Input unit" is defined in the MD:
Instead of rev/min, it can also be 0.1 rev/min

The **"workpiece-related actual value system"** can be activated or deactivated as follows:



You select the appropriated input field using the cursor keys.

You activate or deactivate the function each time the PLUS/MINUS key is operated.The input field shows you whether the function is active or not.

Workpiece-related actual value system  = not active

Workpiece-related actual value system  = active

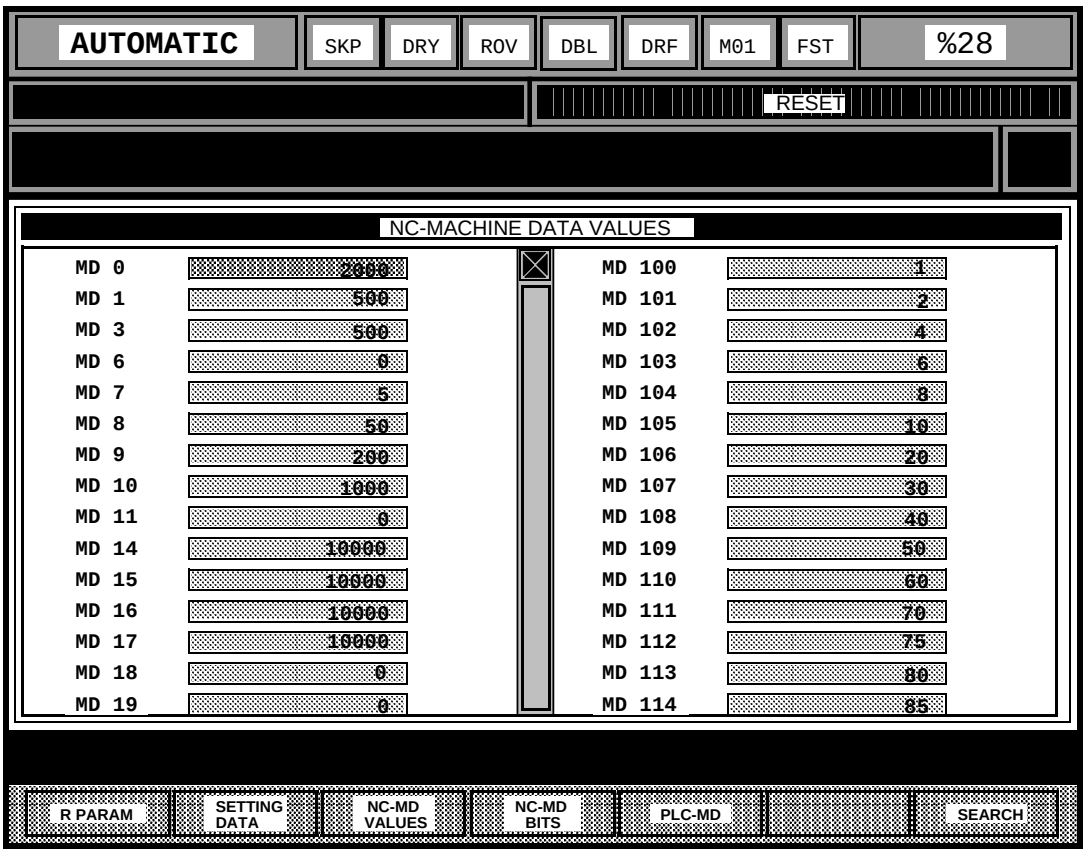
Note:

The field situated above left is  selected by operating the HOME key. (Min. working area X-axis).

© Siemens AG 1990 All Rights Reserved

4.4.3 NC-MD values

The following display appears after the data type NC-MD VALUES has been selected using the relevant softkey.



Note:

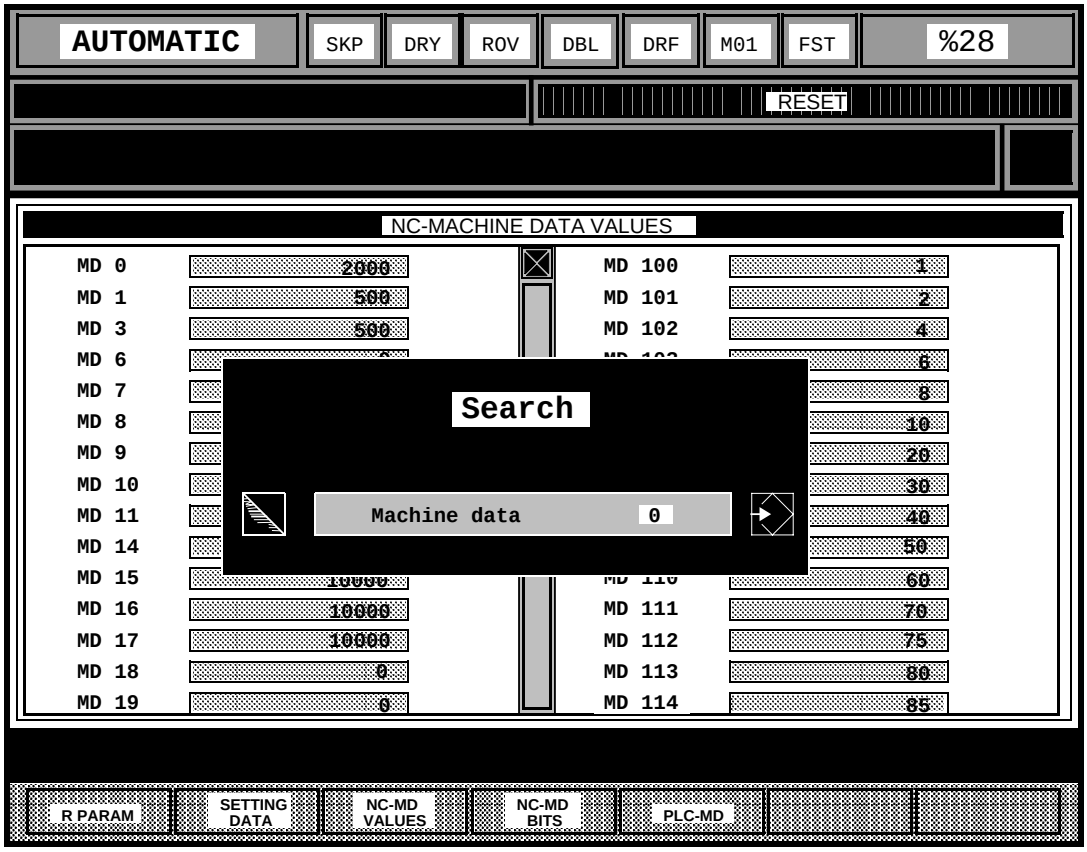
A password is requested when NC-MD values are entered or altered.

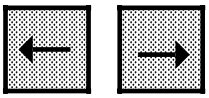
© Siemens AG 1990 All Rights Reserved

Sequence of operation when selecting MC-MD values using the SEARCH softkey

SEARCH

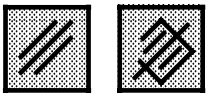
Operate the SEARCH softkey to open the following window:





Now key in the required NC-MD number in the input field which has already been opened, using the numeric keys.

Use the cursor and edit keys where necessary.

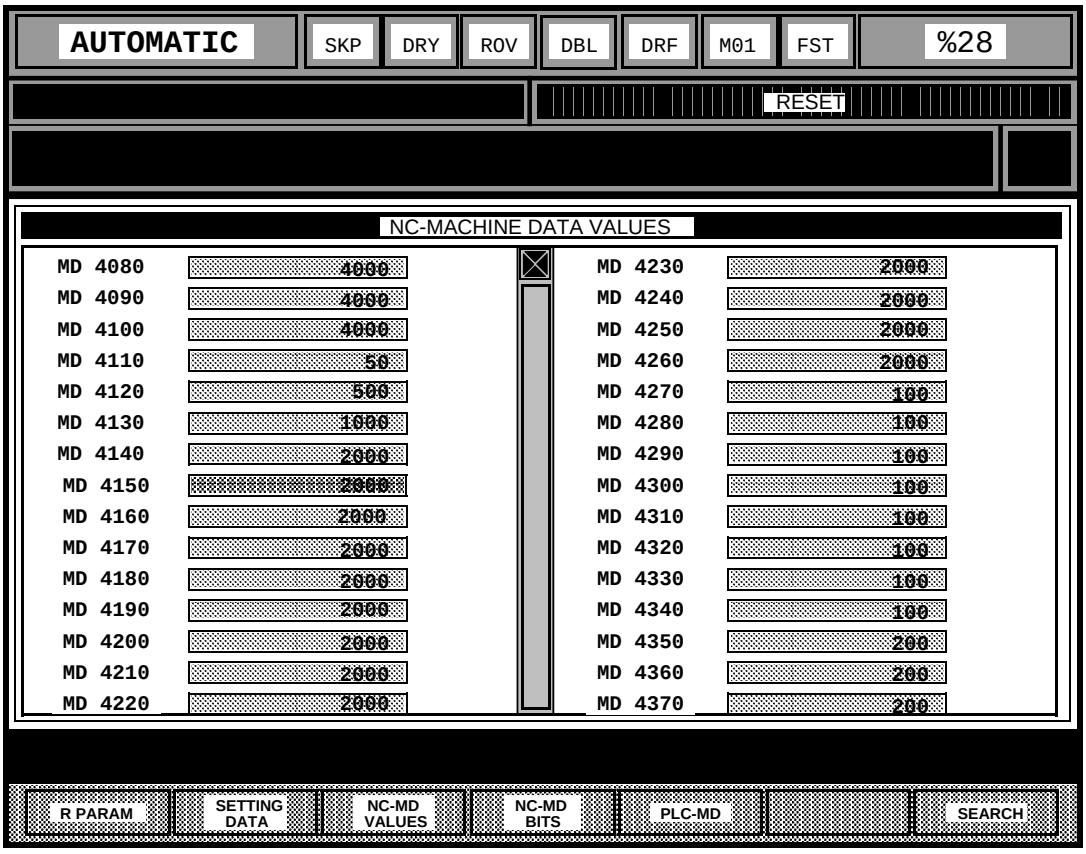


Operating the INPUT key enters the value and simultaneously closes the window.

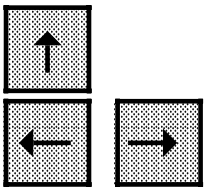
© Siemens AG 1990 All Rights Reserved

Example:

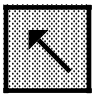
When NC-MD 4150 has been selected, the following display appears:



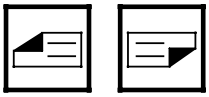
Sequence of operation for manual selection of an NC-MD value



You shift the dark input field within the whole NC-MD value area using these keys.



You shift the input field to the NC-MD value situated above left by operating the HOME key.



You page up or down one display at a time using these keys.

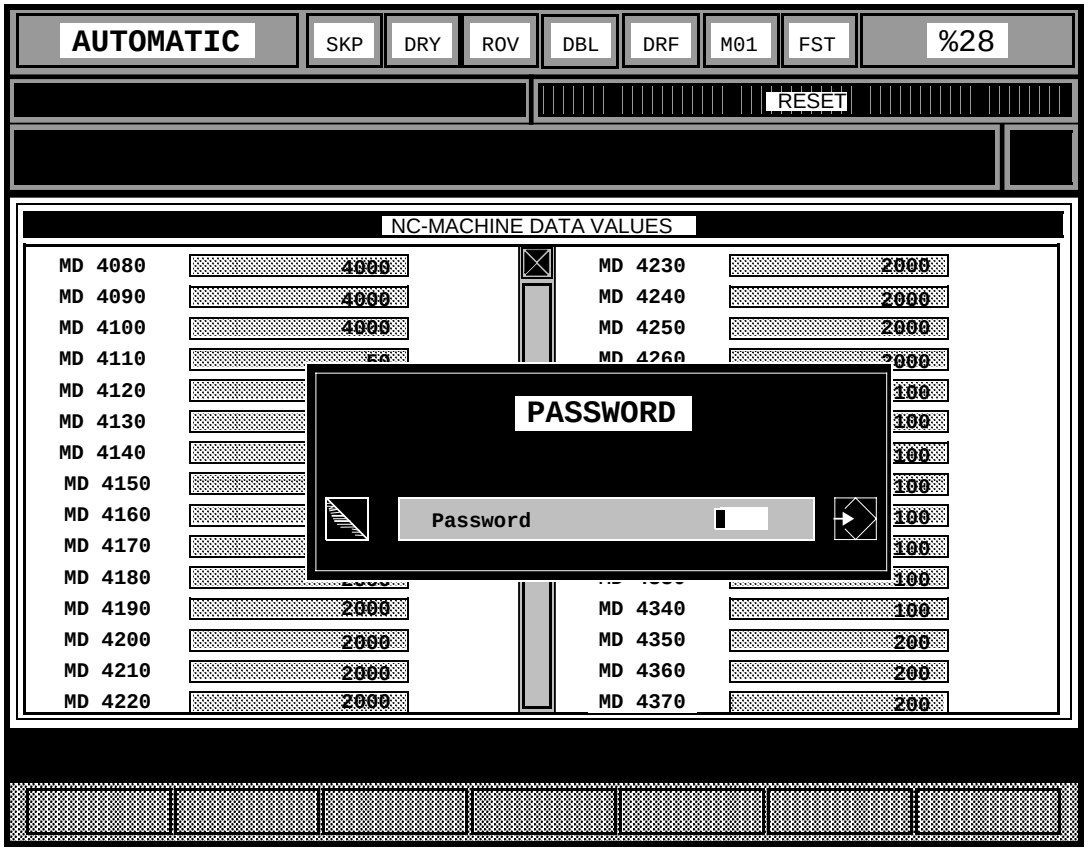
© Siemens AG 1990 All Rights Reserved

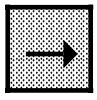
Sequence of operation to alter an NC-MD value

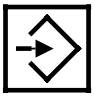
Prerequisite:

You have selected the NC-MD manually or by means of the SEARCH softkey.
The password has not been entered.

You open the following window when you operate the EDIT key:



Now key in the password for the control in the input field which has already been opened.

Use the cursor and edit keys where necessary.

The keyed in password is checked using the INPUT key. If the password is not correct, the window remains open. Enter a new password. The window will close if the password is correct.
The NC-MD values can now be altered as described in Section 2.4.4 (Entry in input fields).

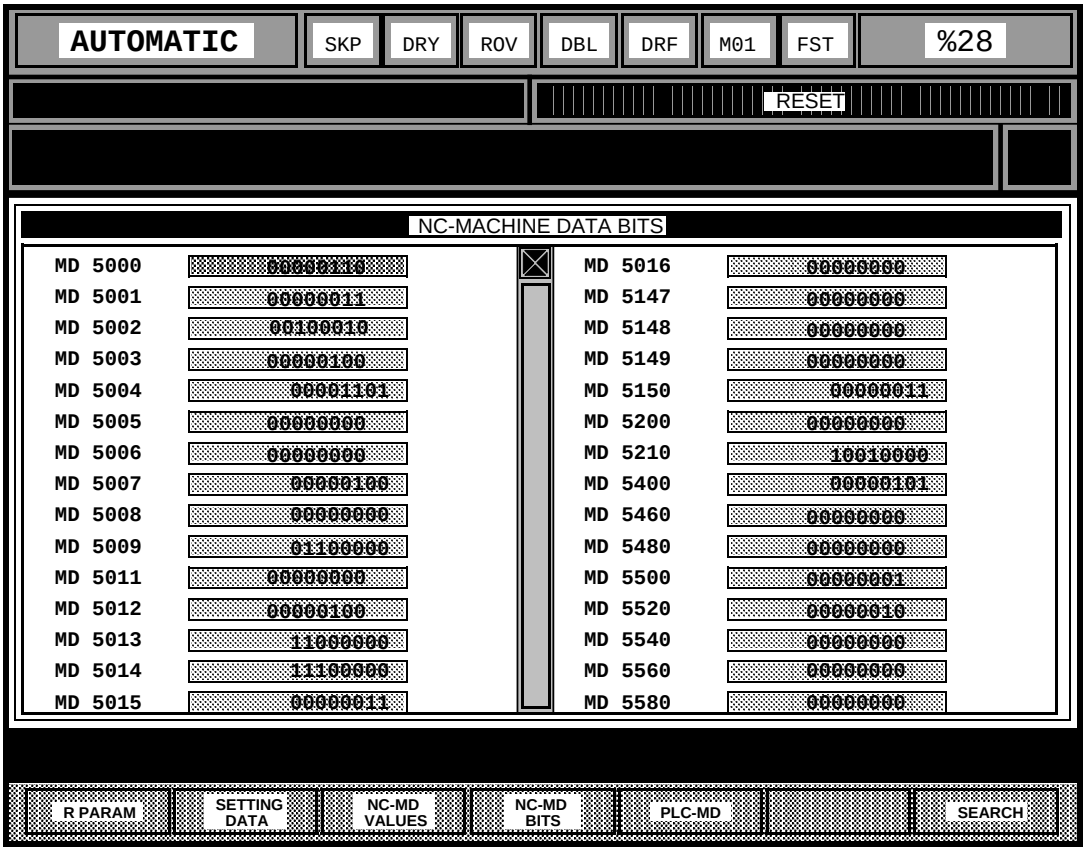
© Siemens AG 1990 All Rights Reserved

Notes:

- When one password is entered, entries in the data types NC-MD BITS and PLC-MD are also possible.
- the password must be re-entered the next time the control is switched on.
- Effective alterations to the NC-MD values depend on the NC-MD number after POWER ON, RESET, NC-STOP/START or immediately.

4.4.4 NC-MD BITS

The data type NC-MD BITS is selected using the appropriate softkey. The following display appears:



Note:

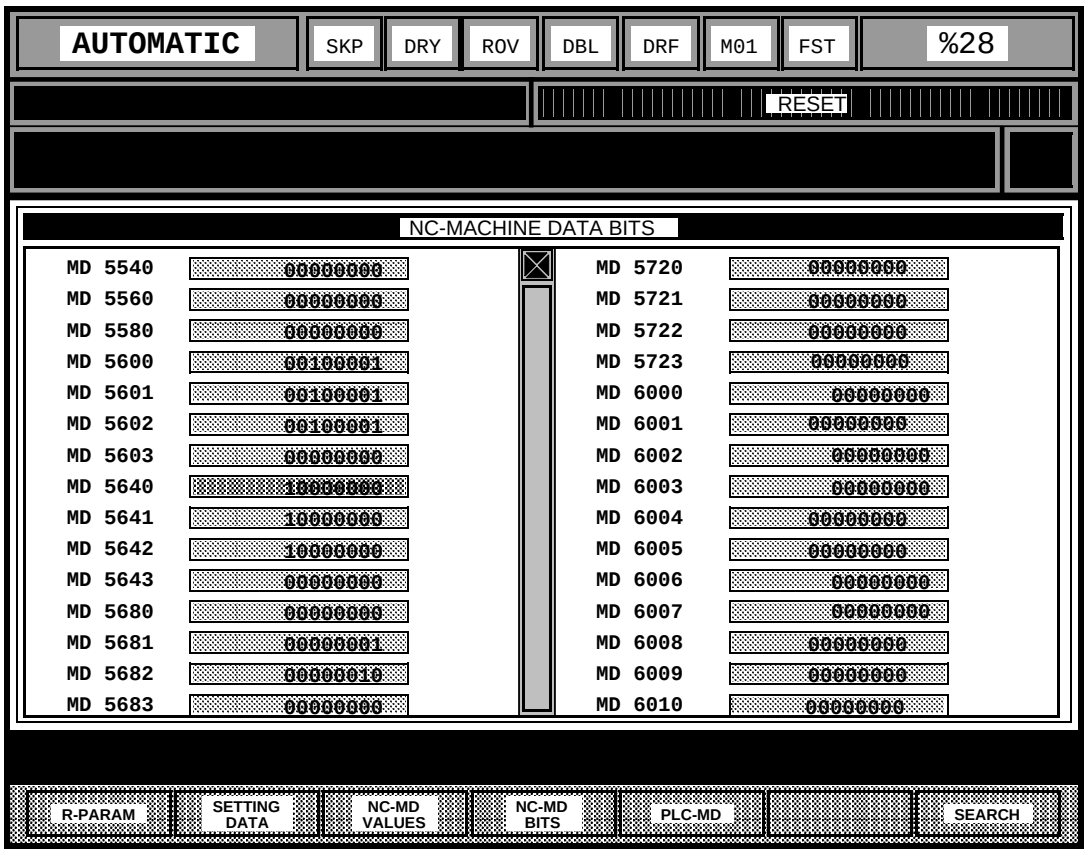
The password is requested when NC-MD BITS are entered or altered.

© Siemens AG 1990 All Rights Reserved

Sequence of operation when selecting an NC-MD bit using the SEARCH softkey

Proceed as described in Section 4.4.3.

If, for example, NC-MD 5640 is selected, the following display appears:



Sequence of operation for manual selection an NC-MD bit

Proceed as described in Section 4.4.3.

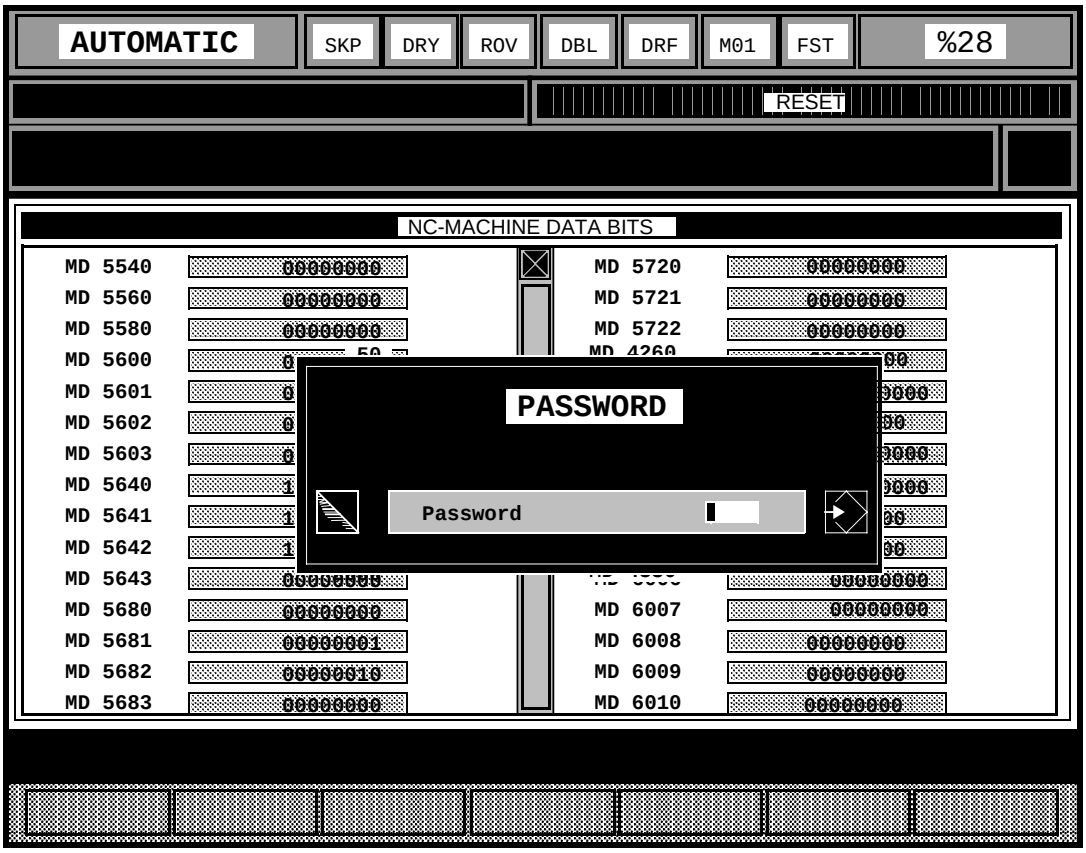
Sequence of operation for altering an NC-MD bit

Prerequisite:

You have selected the input field required manually or using the SEARCH softkey.
"The password has not been entered".

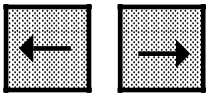
The following window opens when you operate the EDIT key:

© Siemens AG 1990 All Rights Reserved

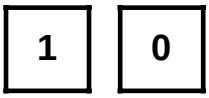


Now key in the password using the numeric keys. Operating the INPUT key enters the password and simultaneously closes the window.

Now open the selected input field by operating the EDIT key. The cursor is on bit 7.



Shift the cursor field using these keys onto the bit to be altered and key in the desired value using the numeric keys 0 or 1.



Operating the EDIT key again closes the input field. The original bit pattern remains.

After input of all bits to be altered, enter bit pattern using the INPUT key.

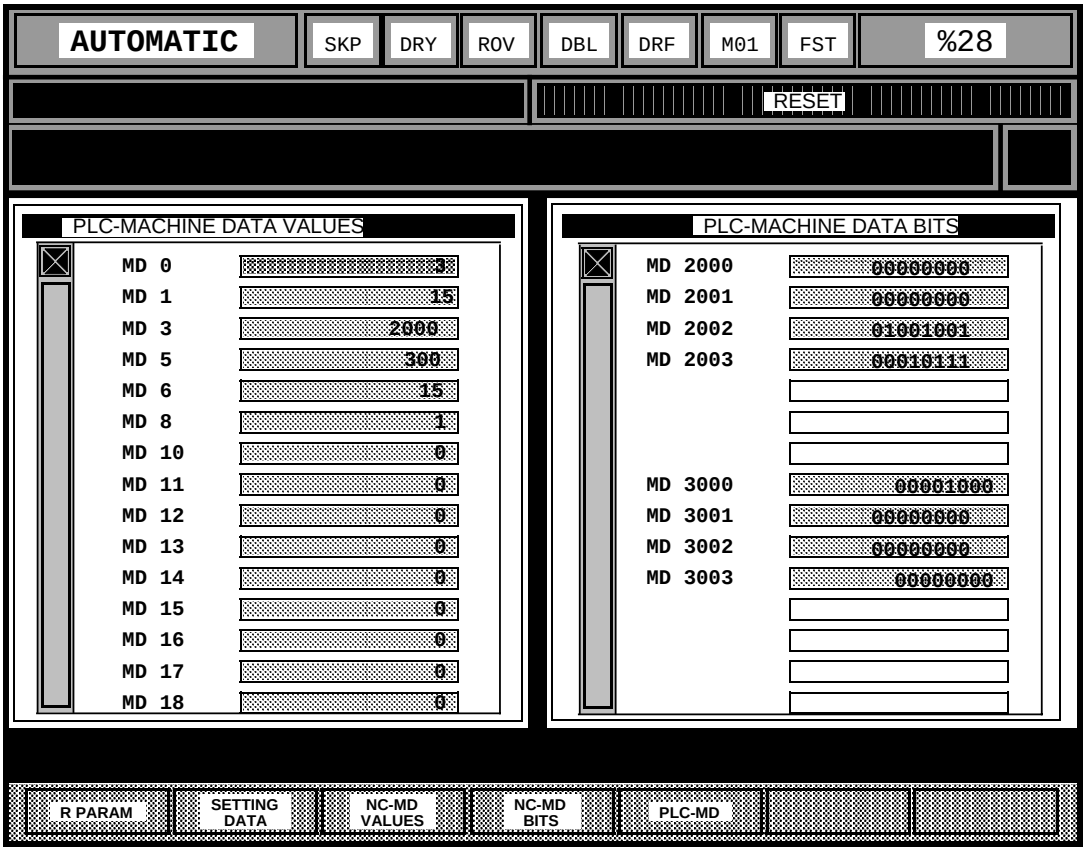
© Siemens AG 1990 All Rights Reserved

Notes:

- If the password is only entered once, it is also possible to make entries in the NC-MD VALUES and PLC-MD data types.
- The password must be re-entered when the control is switched on again.

4.4.5 PLC-MD

Select the data type PLC-MD using the appropriate softkey. The following display appears:



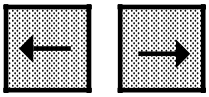
Note:

The password is requested when a PLC-MD is entered or altered.

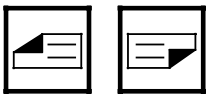
© Siemens AG 1990 All Rights Reserved

Sequence of operation for selecting a PLC-MD

Use these keys to move the darkened input field within the whole MC-MD area.



You move the input field to the PLC-MD situated above left by operating the HOME key.



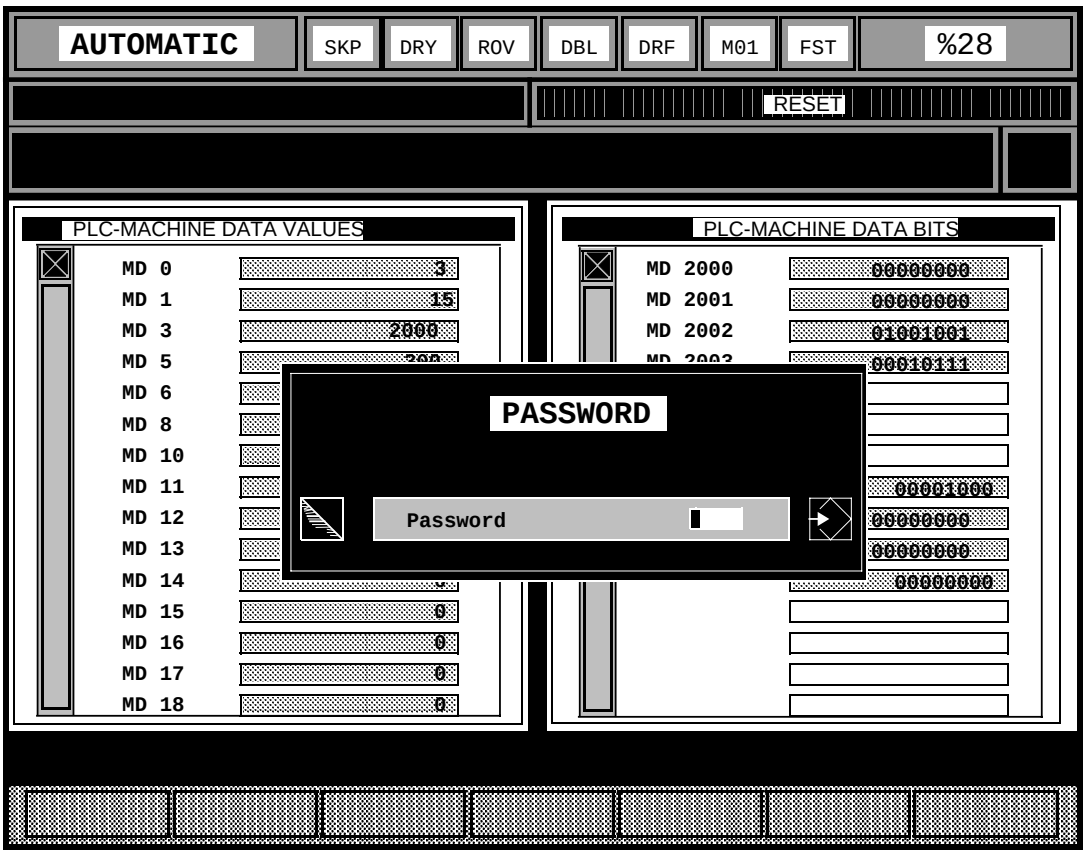
You page one display at a time forwards or backwards using these keys.

Sequence of operation for altering a PLC-MD

Prerequisites: You have selected the appropriate input field manually.

Operate the EDIT key to open the following window:

© Siemens AG 1990 All Rights Reserved



Now key in the password using the numerical keys. By pressing the INPUT key you enter the password and simultaneously close the window.

Operate the EDIT key to open the selected input field.

Now key in the required value using the numerical keys.

0 ... **9**

Pressing the EDIT key again closes the input field. The original value is maintained.

After keying in the value to be altered, operate the INPUT key to enter the value.

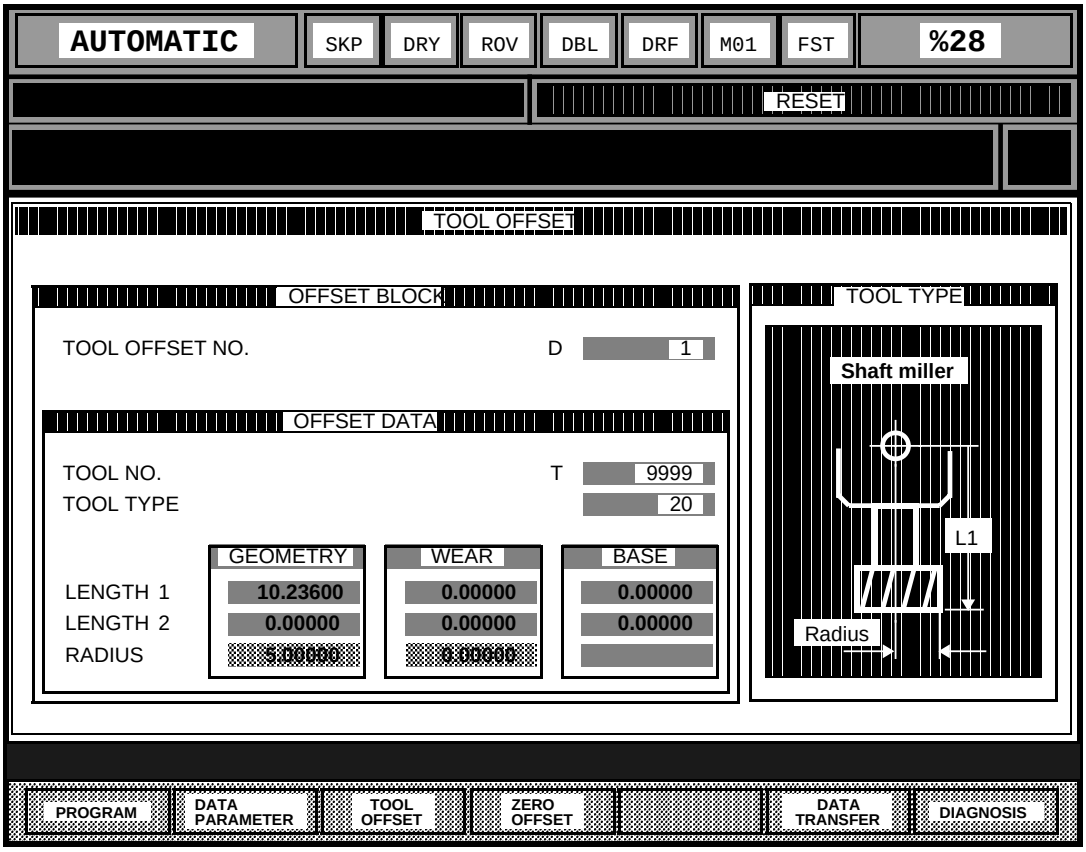
© Siemens AG 1990 All Rights Reserved

Notes:

- If the password is only entered once, it is also possible to make entries in the NC-MD VALUES and PLC-MD bits.
- The password must be re-entered when the control is switched on again.

4.5 Tool offset

By operating the appropriate key in the basic menu of the data area, the following display TOOL OFFSET appears:



Notes:

- The tool offset numbers in SINUMERIK 805 are D1 to D99.
- Each offset number contains the tool parameters P0 to P9 for entering tool identity and data:
  Each of the tool parameters is described in Section 5.2.4.1.
- For each entered tool type a graphic representation appears on which the position of the offset data is shown.

© Siemens AG 1990 All Rights Reserved

4.5.1 Selecting the tool offset numbers

There are two possible ways of selecting the tool offset numbers
a) using PAGE UP/DOWN softkeys

Sequence of operation 1

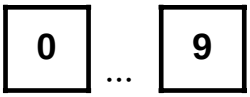
Using the page keys you can move to the next offset number upwards or downwards (D number).

b) using the offset number input

Sequence of operation 2

Use the HOME key to select the OFFSET NO. input field.

Operate the EDIT key to open the input field.



Now key in the required offset numbers using the numerical keys.

After keying in the offset numbers, operate the INPUT key. The parameters of the selected offset numbers are displayed.

4.5.2 Deleting/altering a tool parameter

Sequence of operation

Prerequisite: You have selected a tool offset number for which you wish to delete/alter a tool parameter.

Proceed as described in Section 2.4.4 (Entry in input fields) to delete/alter a tool parameter.

© Siemens AG 1990 All Rights Reserved

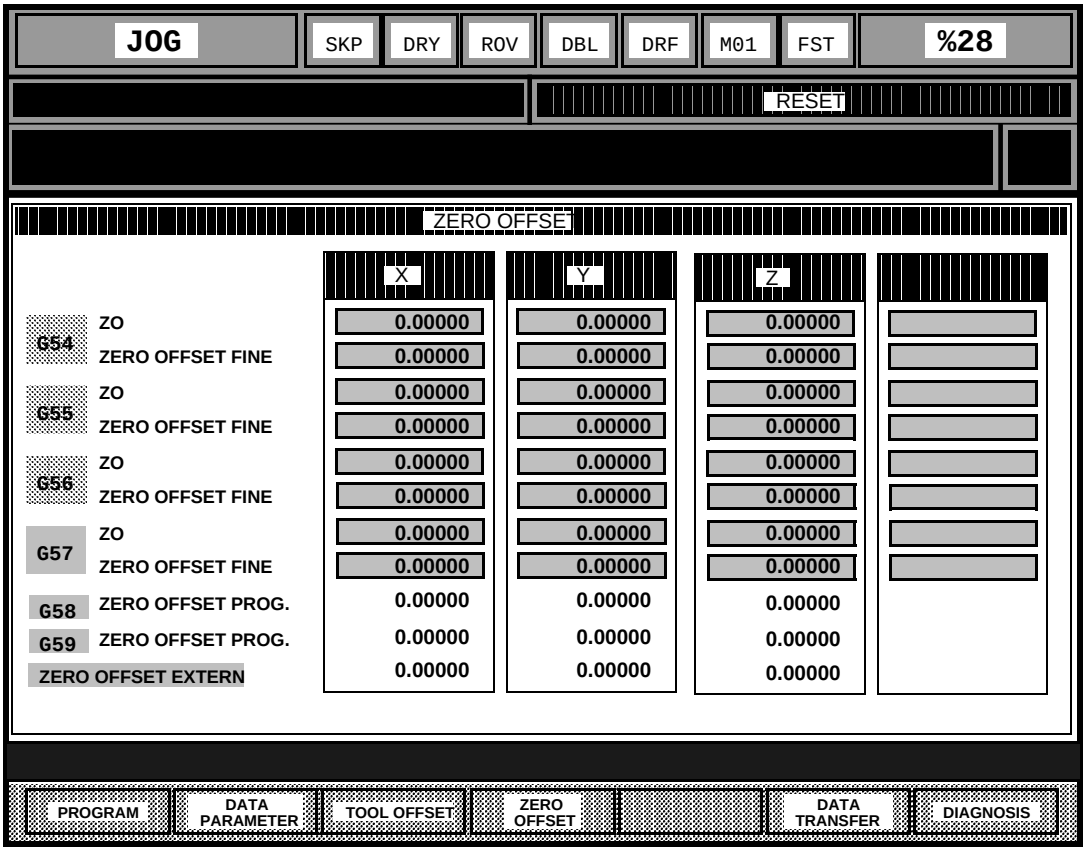
The wear parameters P5 to P7 can be declared invalid by means of machine data!
Additionally, the tool parameters can be extended by the base parameters P8 and P9!
(Please consult the machine tool manufacturer)

Note:

Values in the range +/- 9.999 are possible for wear parameters P5 to P7.

4.6 Zero offset

The following overview of zero offsets for all axes is displayed when the ZERO OFFSET softkey in the basic menu of the data area has been operated:

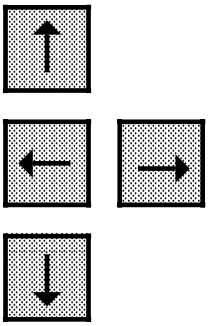


© Siemens AG 1990 All Rights Reserved

Notes:

- Only values for settable zero offsets (G54 to G57) can be entered using the corresponding input fields.
- All zero offsets can be deselected block by block in the part program with G53.

Sequence of operation for entering values for G54 - G57



Use the cursor keys to select the appropriate input field.

You can now enter the required value as described in Section 2.4.4 (Entry in input fields).

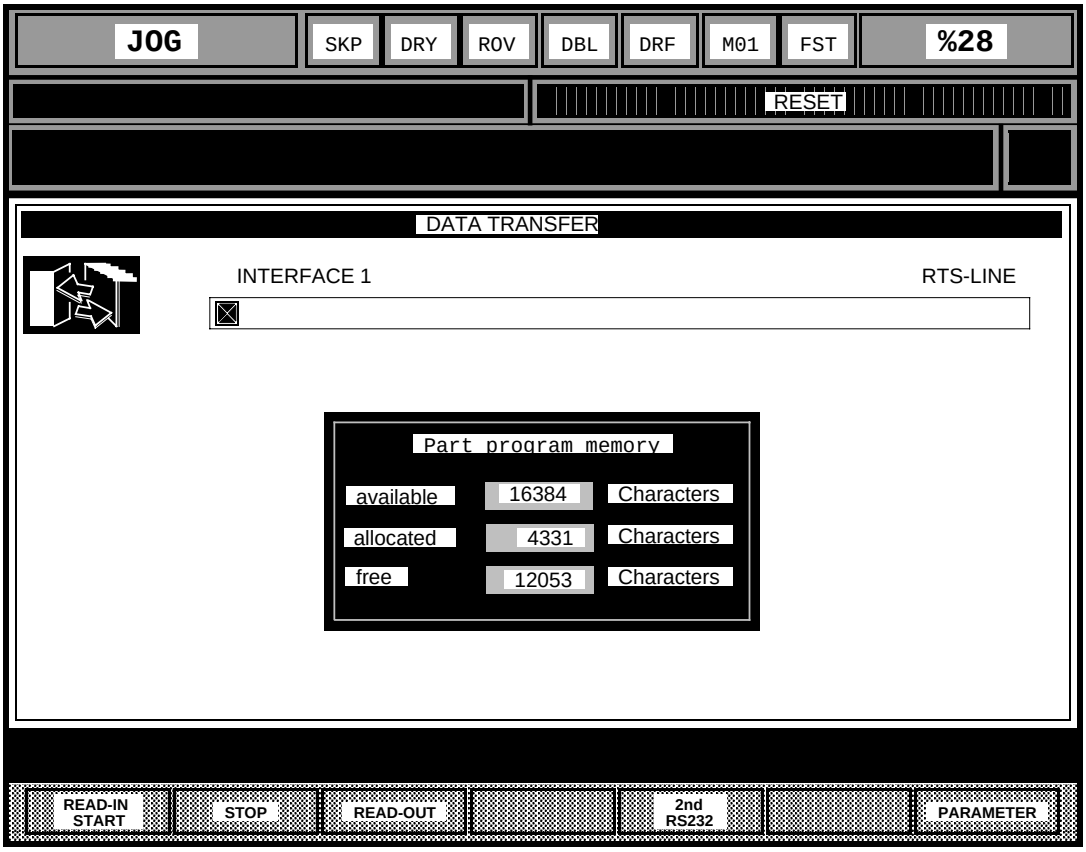
Notes:

- Use the HOME key to select  the original input field (top left).
- The calculator function (Section 2.4.5) can also be applied to all input fields for G54 to G57.

© Siemens AG 1990 All Rights Reserved

4.7 Data transfer

The following display appears when you operate the softkey DATA TRANSFER:



Note:

Available, occupied and free part program memory space is shown in the basic menu DATA TRANSFER.

Explanation of softkeys:

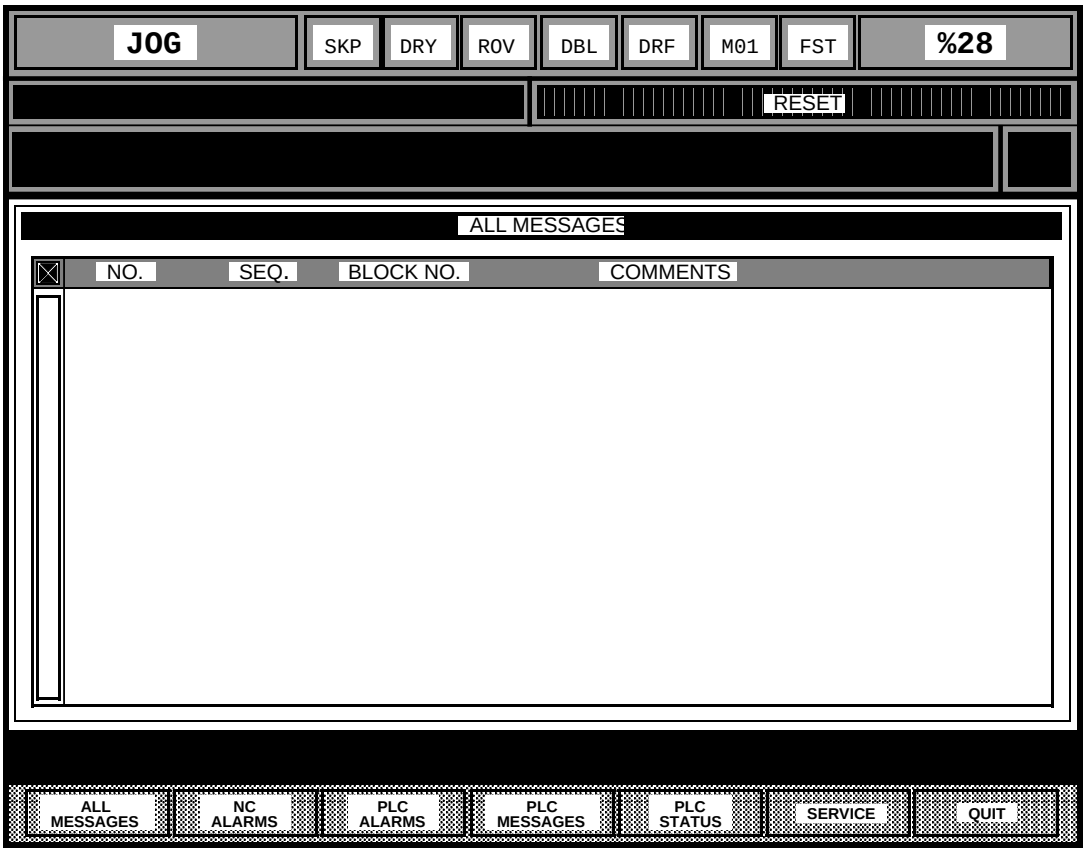
READ-IN START	Prepares the selected interface to receive data.
STOP	Stops data transfer in the read-in/read-out function.
READ-OUT START	Selects the display "data output".
2nd RS232 C	Selection of interface (interface 1 or interface 2 (option)).
PARAMETER	To alter the transfer parameters.

See Section 6, Data Interfaces, for additional information.

© Siemens AG 1990 All Rights Reserved

4.8 Diagnostics

The following display appears when you operate the DIAGNOSIS softkey in the basic menu of the data area:



©Siemens AG 1990 All Rights Reserved

Explanation of softkeys:

ALL MESSAGES	All active NC, PLC alarms and PLC messages are displayed.
NC ALARMS	Only the NC alarms are displayed.
PLC ALARMS	Only the PLC alarms (system alarms and user alarms) are displayed.

Note:

See Section 8.3.2 for further details regarding alarms/messages.

PLC MESSAGES	Only the PLC user messages are are displayed.

Note:

PLC user alarms and messages must be stored by the manufacturer as a subroutine (see Programming Guide Section 1.6).

PLC STATUS	State of inputs, outputs, flags etc in the PLC can be scanned using the PLC STATUS function (Section 4.8.1).
SERVICE	This key is operated by service personnel for an information diagram of the axes and spindles (see Section 4.8.2).

©Siemens AG 1990 All Rights Reserved

4.8.1 PLC status

The PLC STATUS softkey is operated to receive information on the current state of the following memory locations in the PLC and change them where necessary:

Inputs	Input bit	(Ix)	Input byte	(IBx)	Input word	(IWx)
Outputs:	Output bit	(Qx)	Output byte	(QBx)	Output word	(QWx)
Flags:	Flag bit	(Fx)	Flag byte	(FBx)	Flag word	(FWx)
Times:	Time	(Tx)				
Counter:	Counter	(Cx)				
Data:	Data block	(DBx)	Data word	(DWx)		

Operand	E.g.	Read	Write	Format	Value	Area
Inputs		Yes	Yes			0-127
	I 0.0			B	0	
	IB 2			B	0101 1010	
				H	5A	
				D	90	
	IW 2			B	0101 1010 1100 0011	
				H	5AC3	
				D	23235	
Outputs		Yes	Yes			0-127
	Q 0.1			B	1	
	QB 20			B	1101 0110	
				H	D6	
				D	214	
	QW 20			B	1101 0110 1100 0011	
				H	D6C3	
				D	40379	
Flags		Yes	Yes			0-255
	M 0.7			B	0	
	MB 60			B	1011 0100	
				H	B8	
	MW 60			D	180	
				B	1011 0100 0100 0000	
				H	B880	
				D	47232	
Times		Yes	No			0-31
				B		
				H		
				D		

Formats: B=Binary
H=Hexadecimal
D=Decimal

©Siemens AG 1990 All Rights Reserved

Operand	E.g.	Read	Write	Format	Value	Area
Counter		Yes	No			0-31
				B H D		
Data block **Data word**		Yes	Yes			0-255 0-255
	DB 3 DW 9			H D B	A 10 0000 0000 0000 1010	

Formats: B=Binary
H=Hexadecimal
D=Decimal

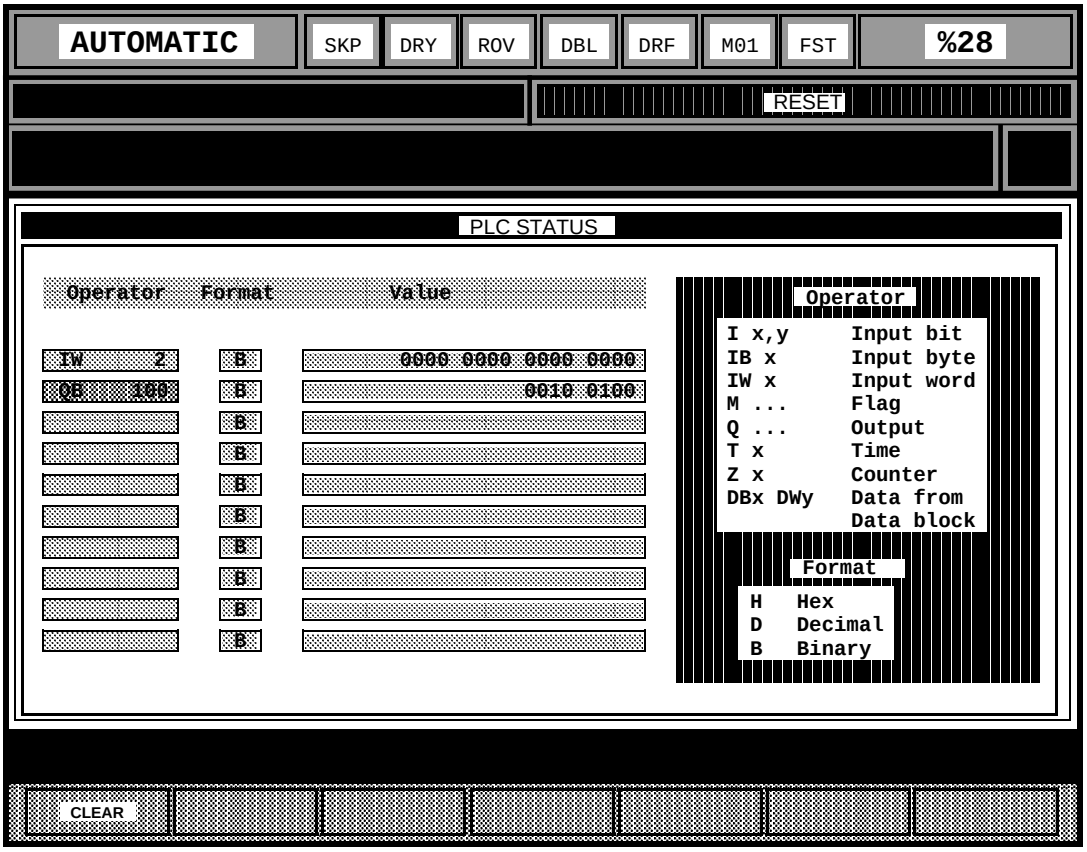
Note:

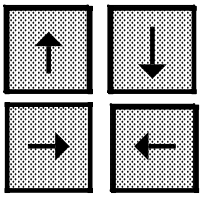
A maximum of 10 operands can be displayed at the same time.

©Siemens AG 1990 All Rights Reserved

Sequence of operation for selecting operands

Prerequisite: PLC status has been selected (Section 4.8).





Use the cursor keys to position the cursor on one line. The required operator (e.g. IW 2 ≙ input word 2) is entered in the operator input field as described in Section 2.4.4 (Entry in input fields).
The required numerical representation is now entered in the format line as described in Section 2.4.4. A separate numerical representation can be selected for each operand or each line.
The operator status is shown in the ”Value” field.

Note:

The increment/decrement keys can be used to run through an operator field or the format in ascending or descending order and select the desired setting.

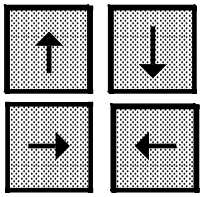
Example:

The cursor is situated in the "Operator" column on an occupied line, which in this case is QB 100. The operator is increased to QB 101 by operating the increment key.

©Siemens AG 1990 All Rights Reserved

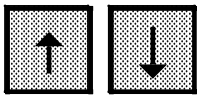
Sequence of operation for altering a value (control)

Prerequisite: Operand has been selected.



Using the cursor keys the cursor is positioned on the "Value" field of the operand to be altered.
The operand contents are now altered in the "Value" field as required, as described in Section 4.2.2 (Entry in input fields).
(Take note of the selected numerical representation).

Sequence of operation for clearing the operand



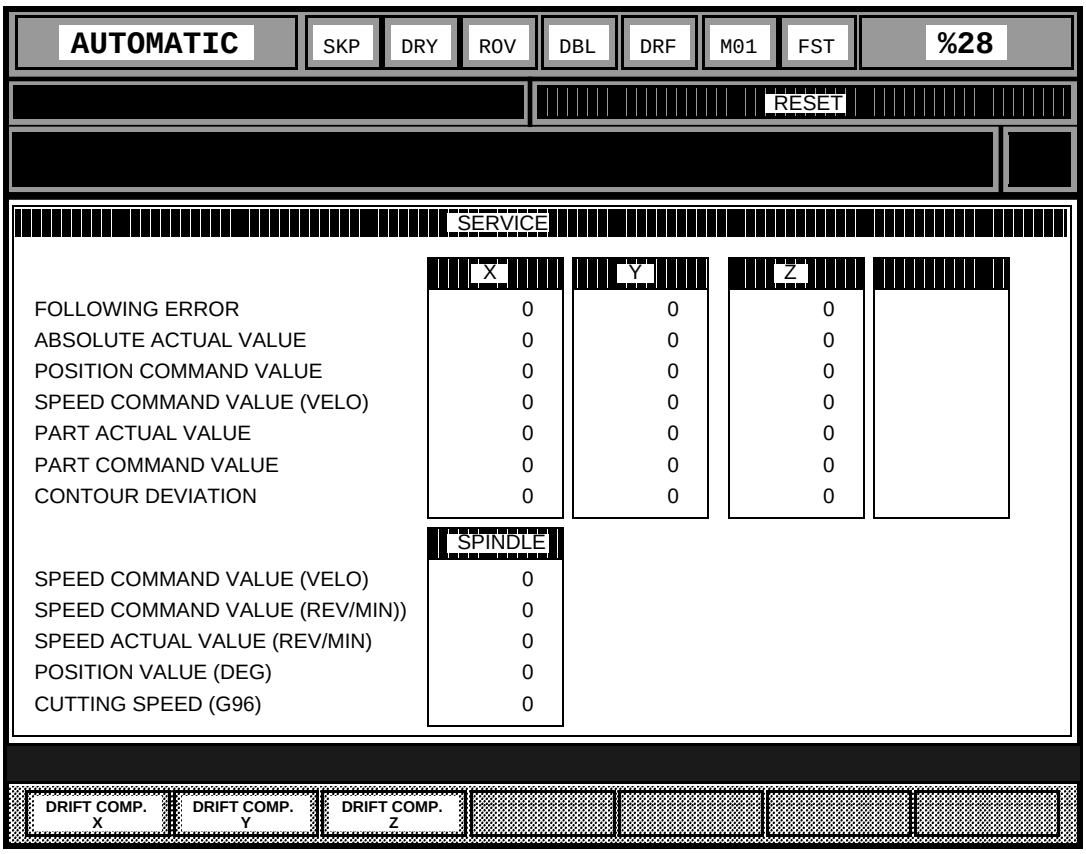
The cursor is moved to the line to be cleared by means of the cursor keys.

Operate the CLEAR softkey. Now the line is without entry.

©Siemens AG 1990 All Rights Reserved

4.8.2 Service

If the SERVICE softkey is operated by service personnel, information regarding the axes and spindles is given.



Explanation of softkeys:

DRIFT COMP. ...	Operating the appropriate softkey executes a drift compensation on the axis concerned.

©Siemens AG 1990 All Rights Reserved

Explanation of service data:

- Absolute actual value

  The actual position of the axis. The position is displayed in the machine reference system (no allowance made for ZO and TO) in position control resolution units.

  Example: If 200 000 is displayed at a position control resolution of $0.5 \cdot 10^{-3}$ mm, this means that the axis is at position 100 (referred to machine zero).

- Following error

  The difference between the position controller command value and the absolute value. The following error is displayed in position control resolution units.

  Example: If 2 000 is displayed at a position control resolution of $0.5 \cdot 10^{-3}$, this means that following error is 1 mm.

- Command value

  This is the programmed target position. When the axis is in the target position (with no movement), the command value is equal to the absolute actual value (if there are any discrepancies, these can be removed by drift compensation). The command value is displayed in position contorl resolution units.

  Example: If 208 000 is displayed at a position control resolution of $0.5 \cdot 10^{-3}$, this means a command value of 104 mm (referred to machine zero).

- Speed command value

  This is the speed command value determined by the control which is supplied to the drive actuator as an analog voltage value.
  Unit: 1.22 mV (≙ 1 VELO)
  8192 VELO ≙ 10 V

  Example: The displayed value 5638 corresponds to a speed command value of 6.87836.

- Part actual value

  The pulses generated by the measuring system multiplied by 4 per position controller sampling interval (5 ms). It is necessary to multiply the number by 4 so that the part actual value (sampling interval 5 ms) can be compared to the part command value (sampling interval 20 ms).
  Unit: position control resolution (standard: 0.5 µm)

  Example: The displayed value 24 corresponds to a 12 µm path every 20 ms at a position control resolution of 0.5 µm.

- Part command value

  Pulses sent to the position control from the interpolator every interpolator cycle (20 ms).
  Unit: position control resolution (standard: 0.5 µm)

  Example: Displayed value 18 corresponds to a 9 µm path every 20 ms at a position control resolution of 0.5 µm.

©Siemens AG 1990 All Rights Reserved

- Contour deviation

  The current contour deviation (fluctuations in the following error caused by adjustments to the speed controller as a result of changes in load).
  Unit: Position control resolution (standard 0.5 µm)

  Example: Displayed value 2 at a position control resolution of 0.5 µm corresponds to a contour deviation of 1 µm.

The following data is displayed in the SINUMERIK 805:

- Speed command value (VELO)

  The command value sent from the control to the spindle actuator in VELO (1 VELO ≙ 1,22 mV).

- Speed command value (rev/min)

  The command value sent from the control to the spindle. The speed command value is displayed in rev/min).

- Speed actual value (rev/min)

  The actual speed of the spindle. The speed actual value is displayed in rev/min).

- Position actual value (degrees)

  Spindle position display in DEGREES. The position is shown in the range 0.1 - 359.9 degrees.

- Cutting speed (m/min)

  Instantaneous cutting speed of tool cutting edge in m/min when G96 has been selected.

©Siemens AG 1990 All Rights Reserved

5 Operating Sequences

5.1 Preliminary remarks

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

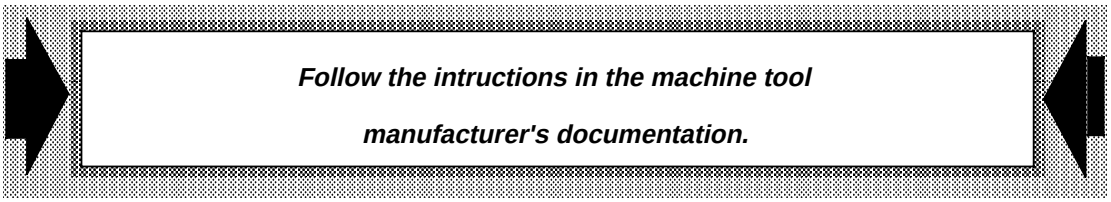
This Section is divided into two parts:

- 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 (operation) the operating methods are explained that occur **during processing**.

5.2 Preparation

5.2.1 Switching on

When you switch the control and machine tool on:



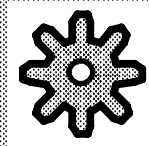
A system screen appears for the first 5 s after switching on the control, after which on the basic menu for the JOG operating mode appears (see also Section 1.2).

© Siemens AG 1990 All Rights Reserved

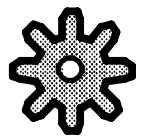
5.2.2 Approach reference point ("REFPOINT")

After switching on, approach the reference points for the various axes. This synchronises the control with the machine. Axes can be referenced in different ways, therefore:

Please follow the machine tool manufacturer's instructions!



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

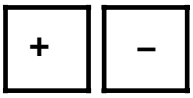


Sequence of operation

Prerequisite: Operating mode JOG has been selected (see Section 3.2.1).

Operating mode JOG-REF is selected using the softkey.

Using the axis selection key on the hand-held unit, select the axis to be referenced (e.g. axis Y).



Now operate the direction keys to commence the reference point approach of the Y axis.

© Siemens AG 1990 All Rights Reserved

Notes:

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

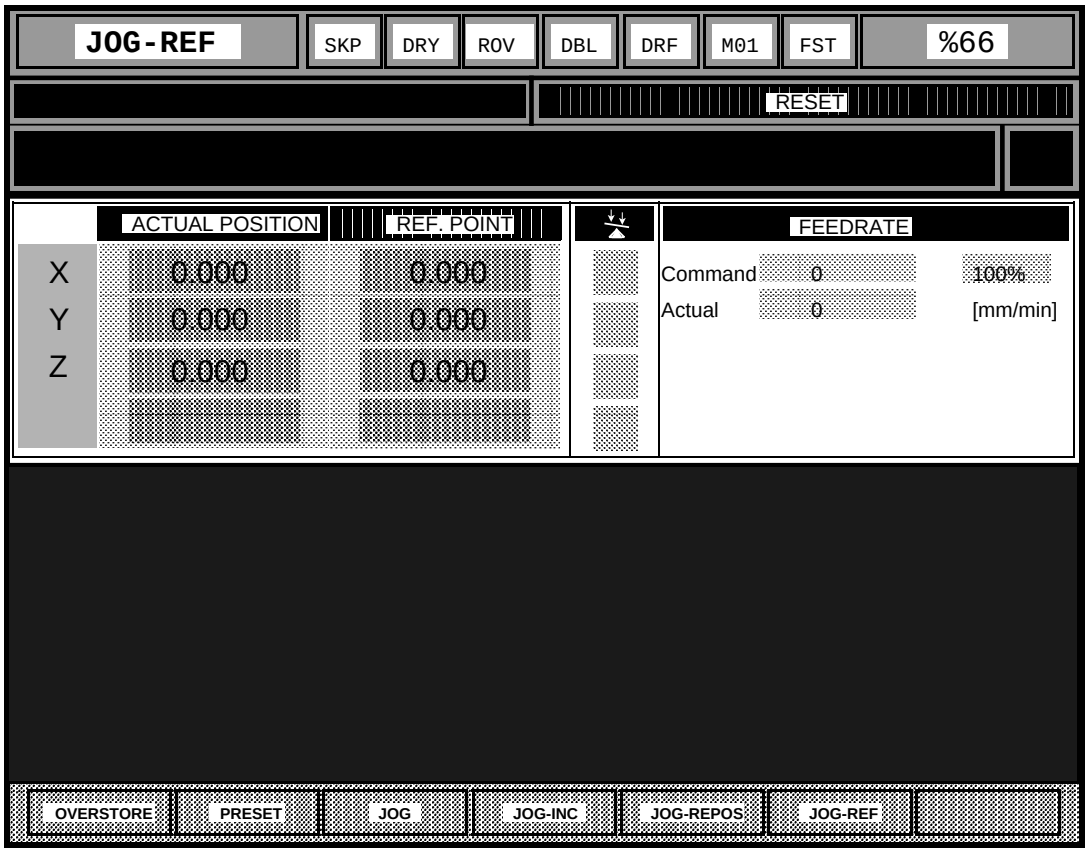
  If you have operated the incorrect direction key (e.g. **+** instead of **−**),

  no traverse motion will result as the operation is rejected.

Using the FEED STOP key on the hand-held unit, you can stop the selected axis before the reference point is reached.
Once the reference point has been reached, the position value (referring to machine zero) is displayed as an actual value on the screen. The corresponding axis name is marked dark on the screen.

The same sequence of operation applies to the other axes. The control is now synchronized with the machine.

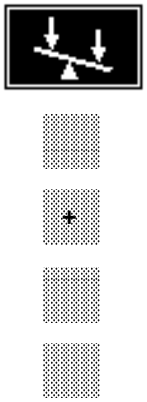
- The reference point approach operation can be carried out simultaneously on a maximum of two axes.



© Siemens AG 1990 All Rights Reserved

5.2.3 Automatic drift compensation

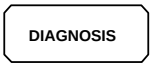
If a plus or minus sign remains in the screen field (see basic display JOG-REF) after the traversing movement of an axis in the JOG mode, for example, the drift has exceeded the permissible value and drift compensation must be carried out.



Sequence of operation

Prerequisite: The machine area has been selected. The control is in the RESET state.

Use the OPERATING AREA key to select the data area.



Operate the DIAGNOSIS and the

SERVICE softkey

Operating the DRIFT COMP. ..., results in the automatic drift compensation of the desired axis.

Notes:

- In the case of an excessive drift, a following error remains for the axis in question. After the drift compensation has been executed, the following error tends to zero.
- The drift values are entered in the following NC-MDs:
  NC-MD 2720 1st axis
  NC-MD 2721 2nd axis
  NC-MD 2722 3rd axis
  NC-MD 2723 4th axis
- Drift values can also be entered manually in NC-MDs 2720 to 2723 (see Section 3.3.3).
- Drift values greater than 500 can no longer be called drift. Alarm 160* is then given (drift too high).

© Siemens AG 1990 All Rights Reserved

5.2.4 Tool compensation

The machine tool manufacturer's documentation should be consulted about tooling!

In the case of SINUMERIK 805, tool offsets can be entered under the tool compensation numbers D1 - D99 and the corresponding WZ parameters P0 - P9. The basic menu TOOL OFFSET can be selected as follows:

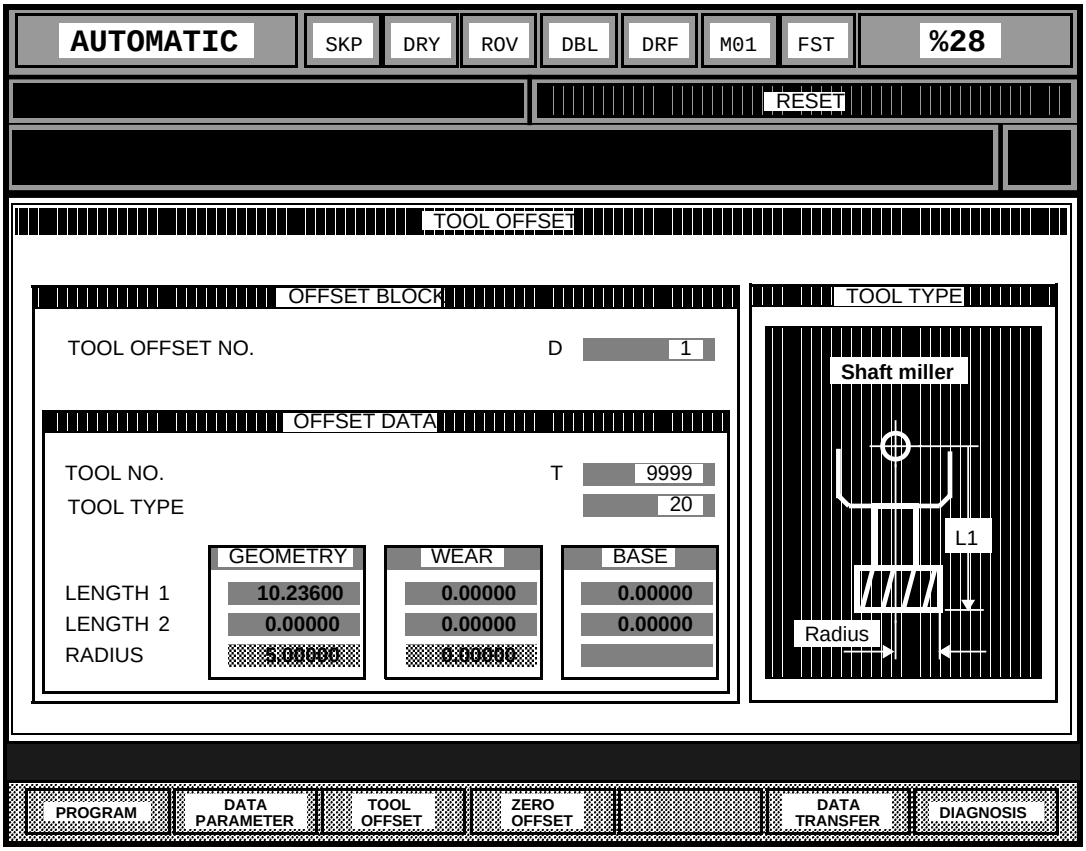
Sequence of operation

Prerequisite: You are in the machine area.

Operate the OPERATOR AREA key to select the data area.

TOOL OFFSET

The following display appears when you operate the TOOL OFFSET softkey:



© Siemens AG 1990 All Rights Reserved

Notes:

- The offset numbers can be selected in two different ways:
  - manually using keys  and  ,

    or

  - by entering the required number in the OFFSET NUMBER input field.

- Each offset number contains the tool parameters P0 to P9 for entering the tool identification and data.

Tool parameter	Designation	Meaning
P0 P1	Tool number Tool type	Tool identity
P2 P3 P4	Length 1 Length 2 Diameter/radius	Geometry parameters
P5 P6 P7	Length 1 Length 2 Diameter/radius	Wear parameters
P8 P9	Length 1 Length 2	Base parameters

The individual parameters are described in later sections.

- The operating sequence for deleting/altering tool paramerers is described in Section 4.5.2.

© Siemens AG 1990 All Rights Reserved

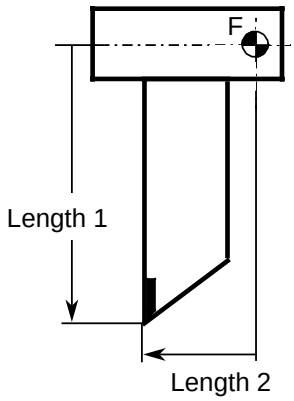
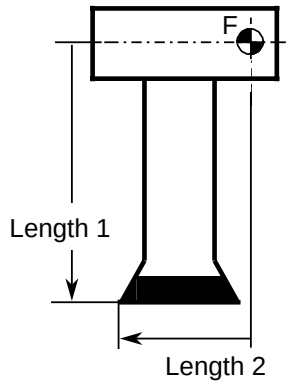
5.2.4.1 Description of tool parameters

Tool parameter P0: Tool number

This input field marked "0" is intended for the input of an 8-digit "tool number". You will not usually have to make such an entry; it may, however, be necessary to enter the number in the case of flexible tool management.

Whether tool numbers are used or not is fixed in the MD!
(See machine tool manufacturer's documentation)

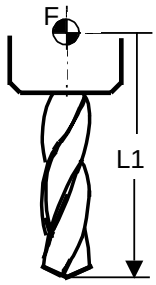
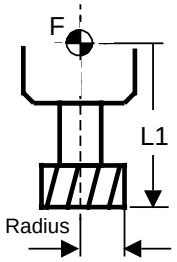
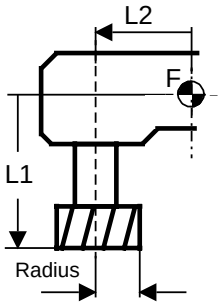
Tool parameter P1: Tool type

Tool		Tool type P1
1. Lathe tool Example: **Turning tool, facing tool**		1 ... 9 **Note:** Input value explanations see: Diagram ”Tool nose centre point positions”
2. Lathe tool Example: **Grooving tool** (cutting edge)		1 ... 9

corresponds to the reference point of the tool clamping device (slide reference point).

© Siemens AG 1990 All Rights Reserved

Tool parameter P1: Tool type

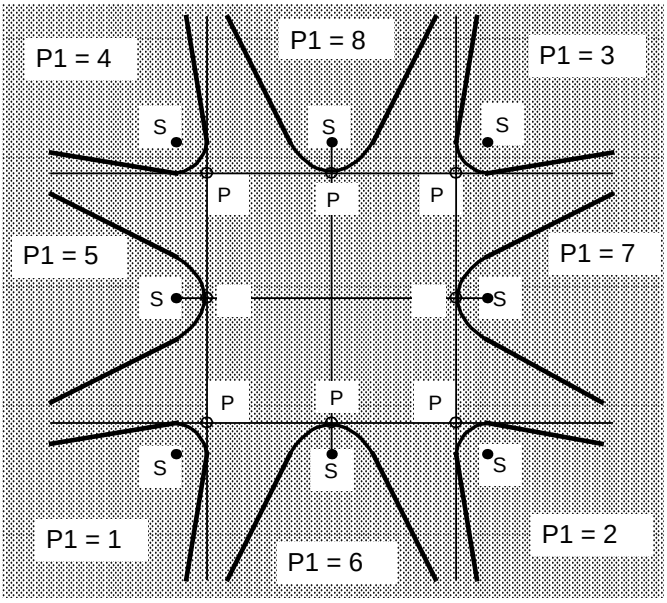
Tool		Tool type P1
3. Tools with length compensation only Example: **Drill**		10
4. Tools with radius compensation and one length compensation Example: **End mill**		20
5. Tools with radius compensation and two length compensations Example: **Angular milling cutter**		30

corresponds to the reference point of the tool clamp (slide reference point).

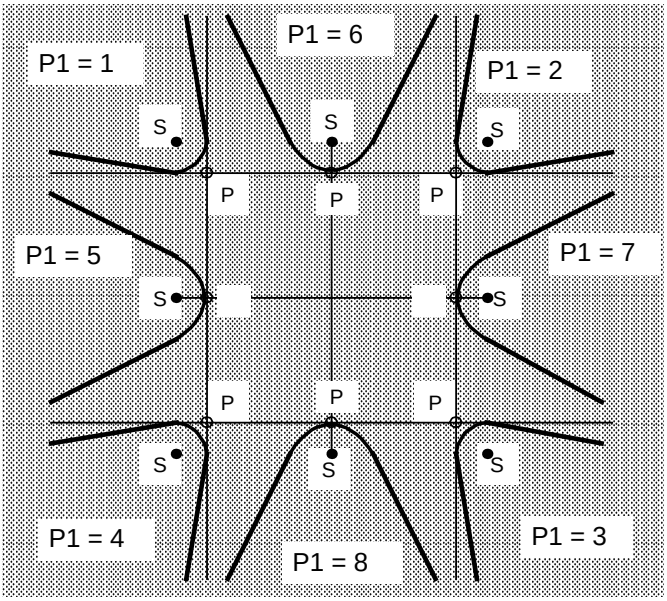
© Siemens AG 1990 All Rights Reserved

Diagram: Tool nose centre point positions

With SINUMERIK 805 (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.

Tool parameter P2: L1 Geometry

Tool parameter P3: L2 Geometry

Tool parameter P4: 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.

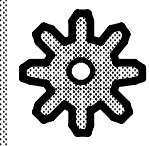
© Siemens AG 1990 All Rights Reserved

Tool parameter P5: L1 Wear

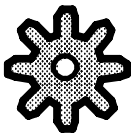
Tool parameter P6: L2 Wear

Tool paramter P7: Diameter/radius

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



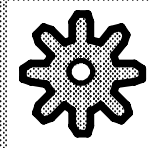
Geometry and wear data input can be inhibited with a key-operated switch. Please see the machine tool manufacturer's documentation!



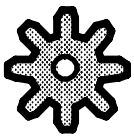
Tool parameter P8: L1 Basic

Tool parameter P9: L2 Basic

Tool parameters "8" and "9" are reserved for special uses. The "basic dimension" permits and additional tool length compensation.



The standard TO parameters can be extended by means of machine data by P8 and P9. Please see the machine tool manufacturer's documentation!



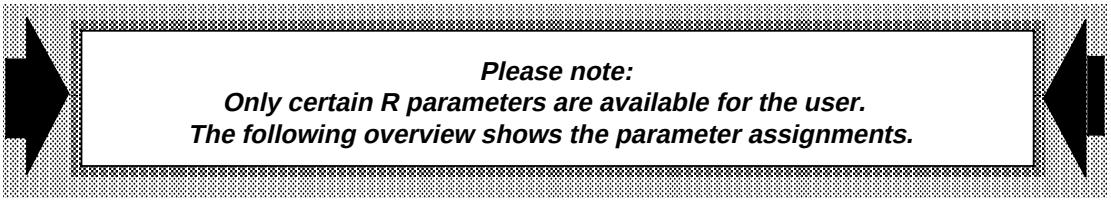
Note:

The input and editing of TO values is described in Section 4.5.

© Siemens AG 1990 All Rights Reserved

5.2.5 R parameters

SINUMERIK 805 recognises R parameters R0 to R999.



R parameter no.	Permanently assigned	Function
R0 : R49	As long as cycles are being processed	Standard cycle transfer parameters
R50 : R99	As long as cycles are being processed	Local R parameters Cycle calculations are carried out with these parameters.
R100 : R109	yes	Reserved for Siemens cycles
R110 : R199	yes	Reserved for measuring cycles. This area is user assignable if no measuring cycles are used.
R200 : R499	yes	These R parameters are used internally (e.g. when using CL800).
R500 : R699		Reserved for Siemens Nuremberg
R700 : R999		User assignable

Part programs can be created in a variety of ways with the help of the R parameters (see SINUMERIK 805 Programming Guide for detailed information).

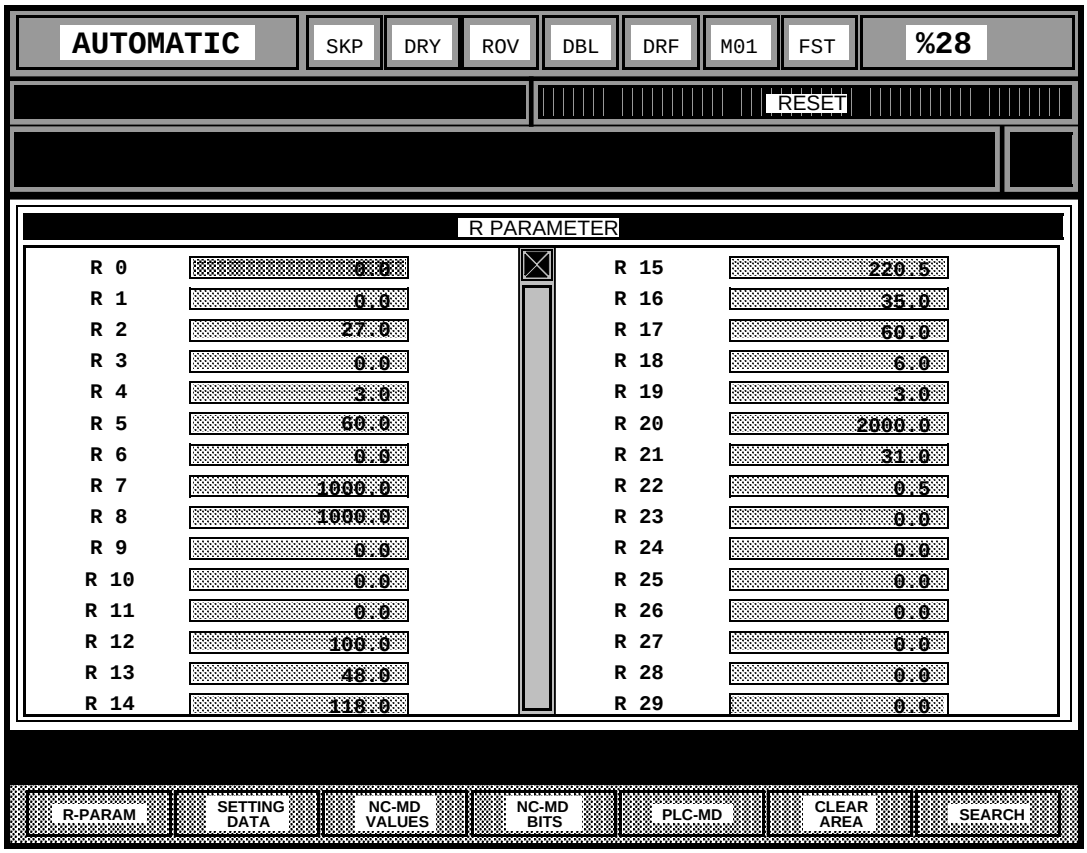
© Siemens AG 1990 All Rights Reserved

Sequence of operation

The SINUMERIK 805 machine area has been selected.

Use the OPERATING AREA key to select the SINUMERIK 805 data area.

The following display appears when you operate the DATA PARAMETER softkey:



Notes:

- Selecting, modifying and clearing R parameters and/or R parameter areas is described in Section 4.4.1.
- The calculator function can also be applied to R parameters.

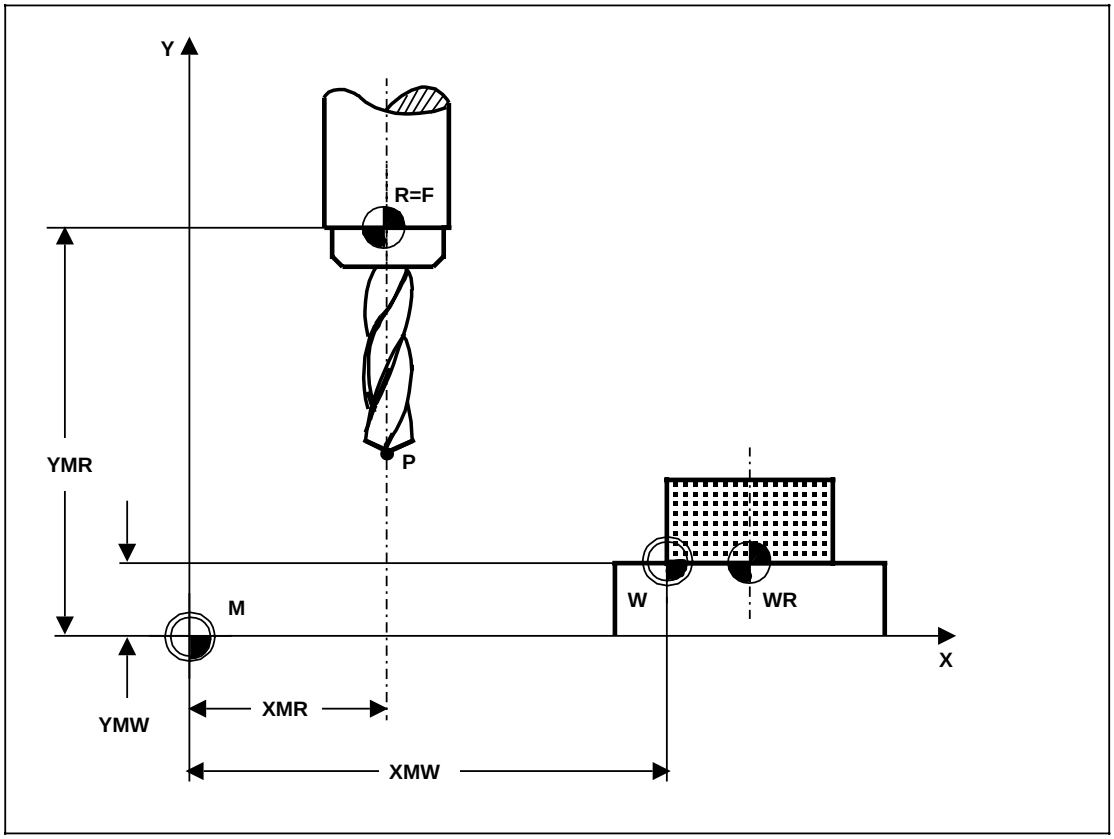
© Siemens AG 1990 All Rights Reserved

5.2.6 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"**.

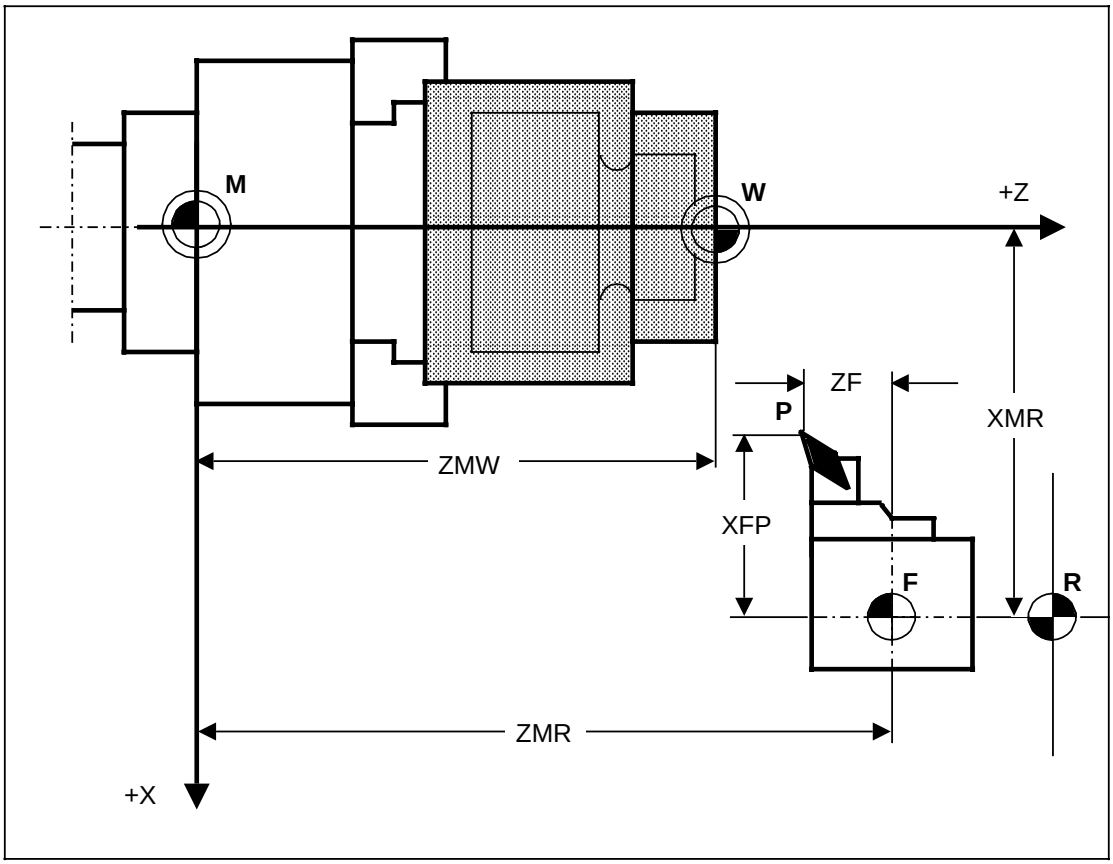
Machine zero point **"M"** and workpiece zero point **"W"** are not identical. The dimensions between the machine zero point **"M"** and the workpiece zero point **"W"** can vary, depending upon the type and fixing of the workpiece. This ***zero offset*** is allowed for during program operation.



Zero offset with a milling machine

P	Tool setting point
M	Machine zero point
W	Workpiece zero point
R	Machine reference point
F	Slide reference point
WR	Workpiece reference point
XMR, YMR	Reference point coordinates
XMW, YMW	Zero offset

© Siemens AG 1990 All Rights Reserved



Zero offset with a turning machine

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

© Siemens AG 1990 All Rights Reserved

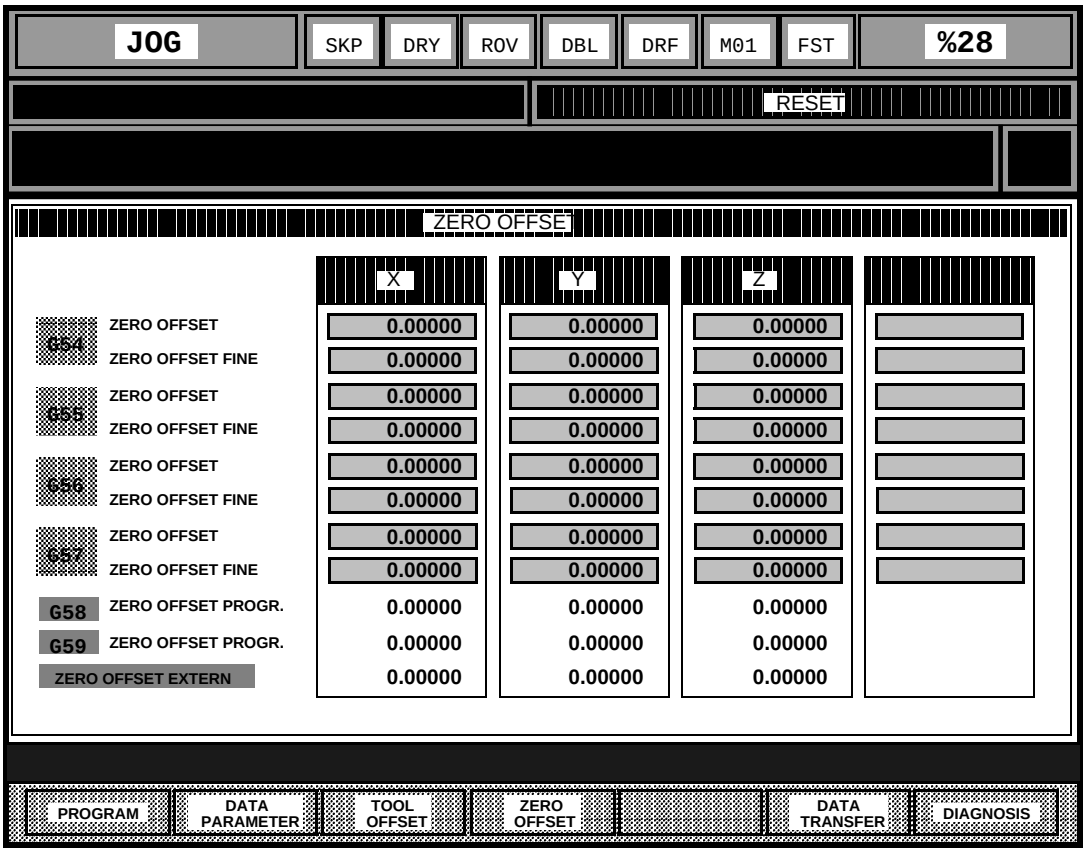
5.2.6.1 Selecting zero point offsets

Sequence of operation for selecting zero point offsets

Prerequisite: The SINUMERIK 805 machine area has been selected.

Use the OPERATING AREA key to select the data area.

The following display appears when you operate the ZERO OFFSET softkey:



The zero offset affecting one axis is the sum of the following zero offsets:

- Settable zero offset (G54, G55, G56 or G57)
- Programmable zero offset (G58 and/or G59)
- External zero offset (ZERO OFFSET EXTERN)

Settable zero offset

You can activate one of the 4 settable zero offsets in the called part program with G54 to G57. The value of the settable zero offset for each axis is the sum of:

- The coarse offset (ZERO OFFSET) and
- the fine offset (ZERO OFFSET FINE)

Note:

Input of the values for G54 to G57 is described in Section 4.5.

Programmable zero offset

Using "G58" and "G59" you can program an additional zero offset for all existing axes in the called part program. Details of how to program these zero offsets are to be found in the Programming Guide (User Documentation for SINUMERIK 805).

The programmed zero offsets of a current program can be displayed on the screen by selecting ZERO OFFSETS.

Note:

The values of the ZERO OFFSETS programmed with G58 and/or G59 are cleared with program end (M02, M30).

External zero offset

The values for the external zero offset (ZERO OFFSET EXTERN) are transferred from the interface control (PLC) to the NC.

These values can also be displayed on the screen by selecting ZERO OFFSETS.

© Siemens AG 1990 All Rights Reserved

5.2.7 Setting data

Using the setting data you can determine and/or activate
– the working area limitation of the individual axes
– the dry run
– the workpiece related actual value display system
– various spindle data

The setting data are entered by you in the SETTING DATA display.

5.2.7.1 Setting data selection

Sequence of operation for selecting setting data

Prerequisite: The SINUMERIK 805 machine area has been selected.

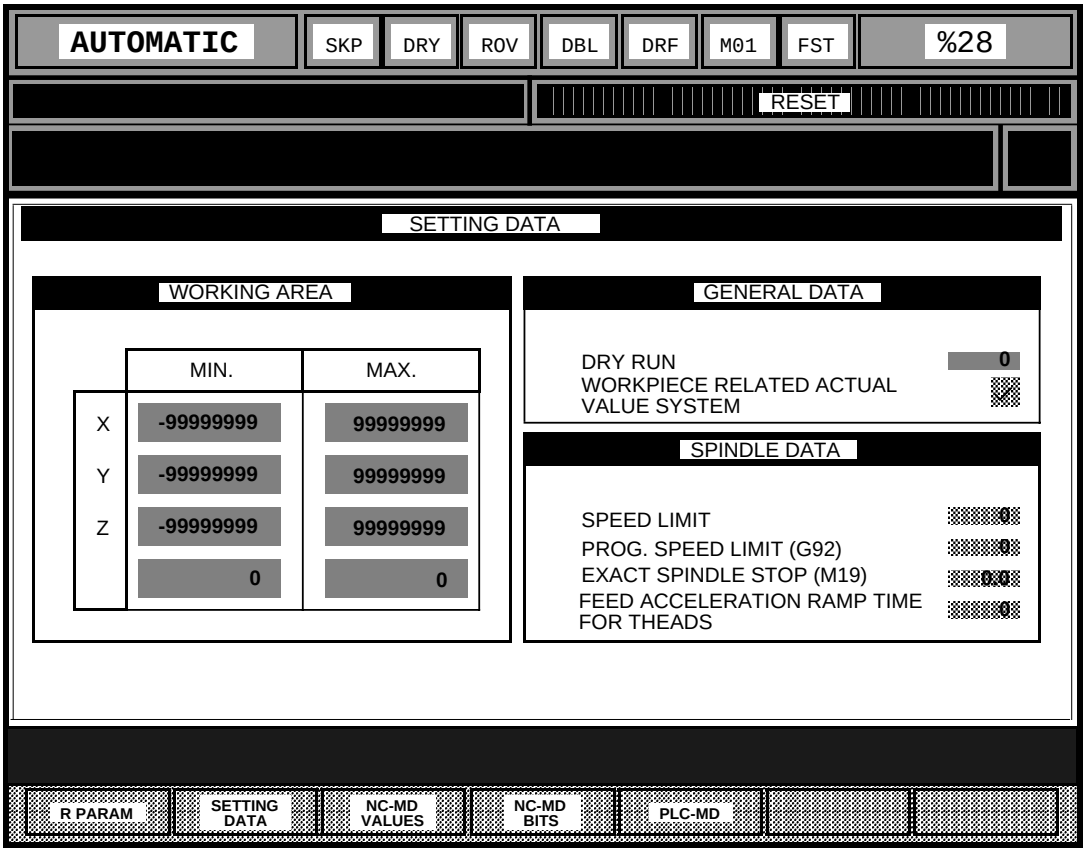
Use the OPERATING AREA key to select the data area.

DATA PARAMETER

Operate the DATA PARAMETER softkey.

SETTING DATA

The following display appears when you operate the SETTING DATA softkey:



© Siemens AG 1990 All Rights Reserved

Values within the following ranges can be entered in the input fields:

Designation	Standard value	Maximum input value	Input unit
Working area limitation			
Min. 1. axis	0	± 99999999	0.001 mm 1)
Min. 1. axis	0	± 99999999	0.001 mm 1)
Min. 2. axis	0	± 99999999	0.001 mm 1)
Min. 2. axis	0	± 99999999	0.001 mm 1)
Min. 3. axis	0	± 99999999	0.001 mm 1)
Min. 3. axis	0	± 99999999	0.001 mm 1)
Min. 4. axis	0	± 99999999	0.001 mm 1)
Min. 4. axis	0	± 99999999	0.001 mm 1)
General data			
Dry run	0	24000	m/min 2)
Spindle data			
Speed limitation	0	12000	Rev/min 3)
Programmed speed limit (G92)	0	12000	Rev/min 3)
Oriented spindle stop (M 19)	0	359.9	1/10 degrees
Feed acc. ramp time for threads	0	5	–

1) The input unit is set in the machine data! It can be 0.0001 mm, 0.001 mm, or 0.01 mm.

2) The input unit is set in the machine data! It can be m/min or mm/min.

3) The input unit is set via machine data! It can be rev/min or 0.1 rev/min (please refer to the machine tool manufacturer's instructions).

Workpiece related actual value system

When the workpiece related actual value system has been selected, the actual value display includes the selected zero offsets and tool offset.
The workpiece zero point is used as the zero point for the actual value display (See also Installation Instructions, Section 11.1).

Note:

The input of values and activation of the "workpiece related actual value system" is described in Section 4.4.2.

© Siemens AG 1990 All Rights Reserved

5.2.8 Program input

Any existing programs (e.g on tape) can be read into the control via RS 232 C interfaces (see Section 6.2).
You can also enter a program using the keyboard.

5.2.8.1 Program input with the keyboard

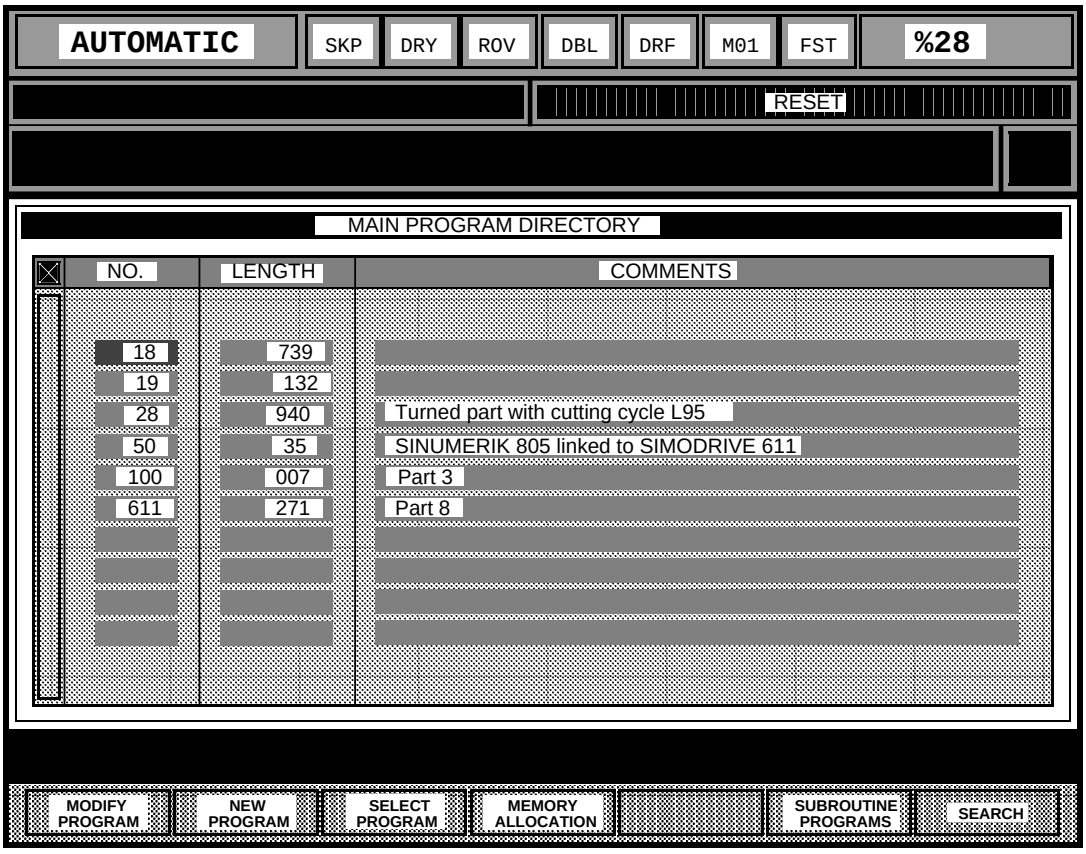
Sequence of operation for selecting the PROGRAM menu

Prerequisite: The SINUMERIK 805 machine area is selected.

Select the data area by means of the OPERATING AREA key.

PROGRAM

The following display appears when you operate the PROGRAM softkey:



Notes:

- Program operation and program editing are described in Section 4.3 (softkey function PROGRAM).
- If the OPERATING AREA softkey is operated again the basic menu of the set operating mode in the machine area appears.

© Siemens AG 1990 All Rights Reserved

5.3 Operation

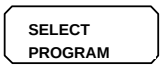
5.3.1 Starting a part program

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.

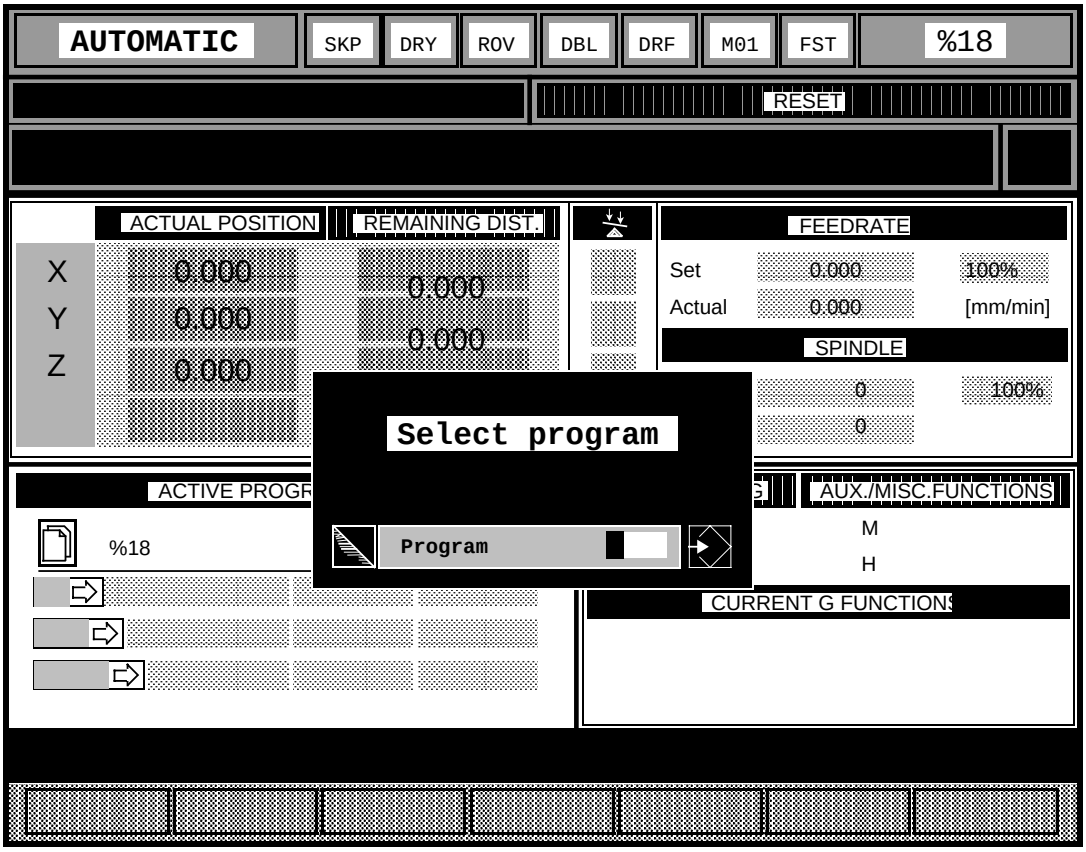
Sequence of operation

Prerequisite: The machine area has been selected.

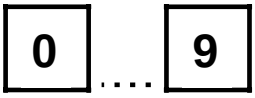
Select the operating mode AUTOMATIC using the appropriate key.



Operate the SELECT PROGRAM softkey.
The following display incorporating an empty input field appears:



© Siemens AG 1990 All Rights Reserved



Now enter the desired program number using the numerical keys.

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

Operate the FEED START and

and

SPINDLE START keys.

You start the program with the NC start key.

Notes:

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

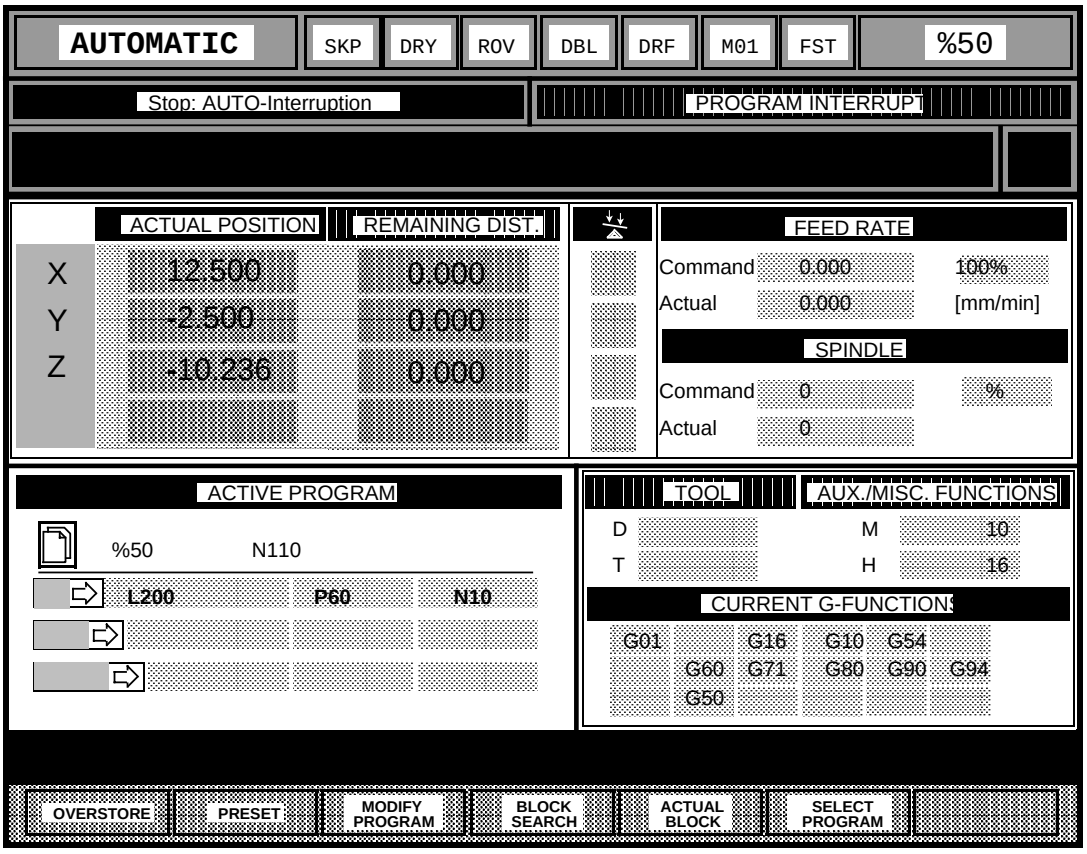
© Siemens AG 1990 All Rights Reserved

5.3.2 Calling ACTUAL PROGRAM or ACTUAL BLOCK in AUTOMATIC mode

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

Sequence of operation

The ACTUAL PROGRAM display has been selected. You switch to the ACTUAL BLOCK display by operating the softkey.



© Siemens AG 1990 All Rights Reserved

Explanation of ACTUAL PROGRAM display

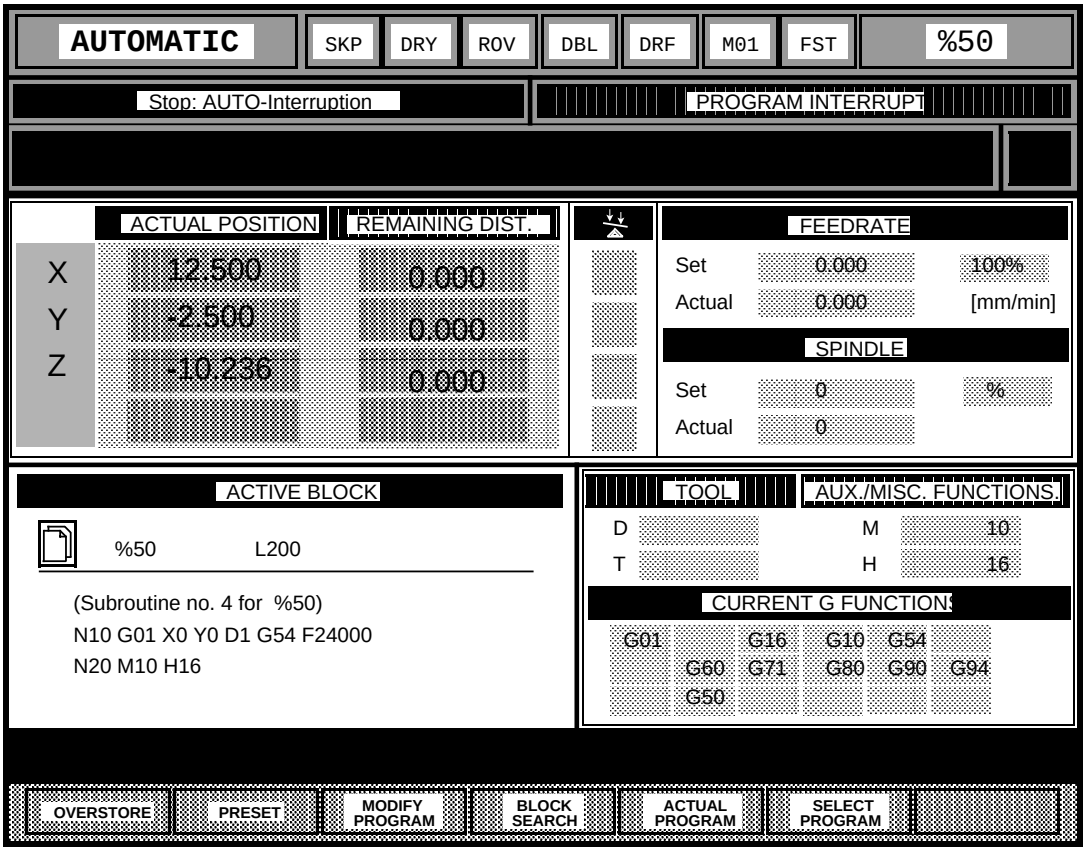
% 50	N110

Selected program display (program pointer) and block number (processing pointer).

1	L200	P60	N10
2			
3			

Display of subroutine number L..., the pass count P... as well as the block number within the subroutine (processing pointer) with three levels of nesting.

The display ACTUAL PROGRAM has been selected. The display is switched over by operating the ACTUAL BLOCK softkey.



© Siemens AG 1990 All Rights Reserved

Explanation of the ACTUAL BLOCK display

% 50	L200		

(Subroutine no. 4 for % 611) LF

N10 610 X0 Y0 D1 654 F24000 LF

N20 M10 H16 LF

The main program or subroutine (3 levels of subroutine nesting possible) belonging to the actual block is displayed in the top line (program pointer).

The following blocks are displayed in the remaining part of the field (6 lines):

- The block **before** the actual block (program comments here)
- The ACTUAL BLOCK itself (Block N10 here)
- The block after the actual block (Block N20 here)

Note:

If lack of space does not permit display of all 3 blocks (block longer than 1 display line), the following are possible:

1. Display of:
   – block before actual block
   – actual block
   – block after actual block (only beginning)
2. Display of:
   – block before actual block
   – actual block
3. Display of:
   – actual block

© Siemens AG 1990 All Rights Reserved

5.3.3 Program interrupt

You can interrupt the program by operating the keys FEED HOLD or SPINDLE HOLD.

Operating sequence

FEED STOP

You operate the key FEED STOP.
The feed drives are brought to a standstill following the programmed paths.

Field FST (FEED STOP) is displayed in reverse video.

Now operate the FEED START key. The feed drives are re-enabled. Program processing continues.

Notes:

FEED HOLD is also displayed when a traverse inhibit prevents the start, e.g.:
- when a control loop fails
- when a monitoring function is operated
- when the feedrate override is at the 0% and while enables apply

SPINDLE STOP

The spindle drive is disabled when the SPINDLE STOP key is operated.

Press the SPINDLE START key to re-enable the spindle drive.
Program processing continues.

© Siemens AG 1990 All Rights Reserved

5.3.4 BLOCK SEARCH function

The BLOCK SEARCH function allows a program start at any point in the part program. During a block search the same calculations are carried out as during normal program operation, the axes, however, do not move.

The output of the auxiliary functions during "Block search" is defined as required using machine data. Depending upon the setting made during installation, the H-, M-, S- and T- functions are output either completely or in part, or completely suppressed.

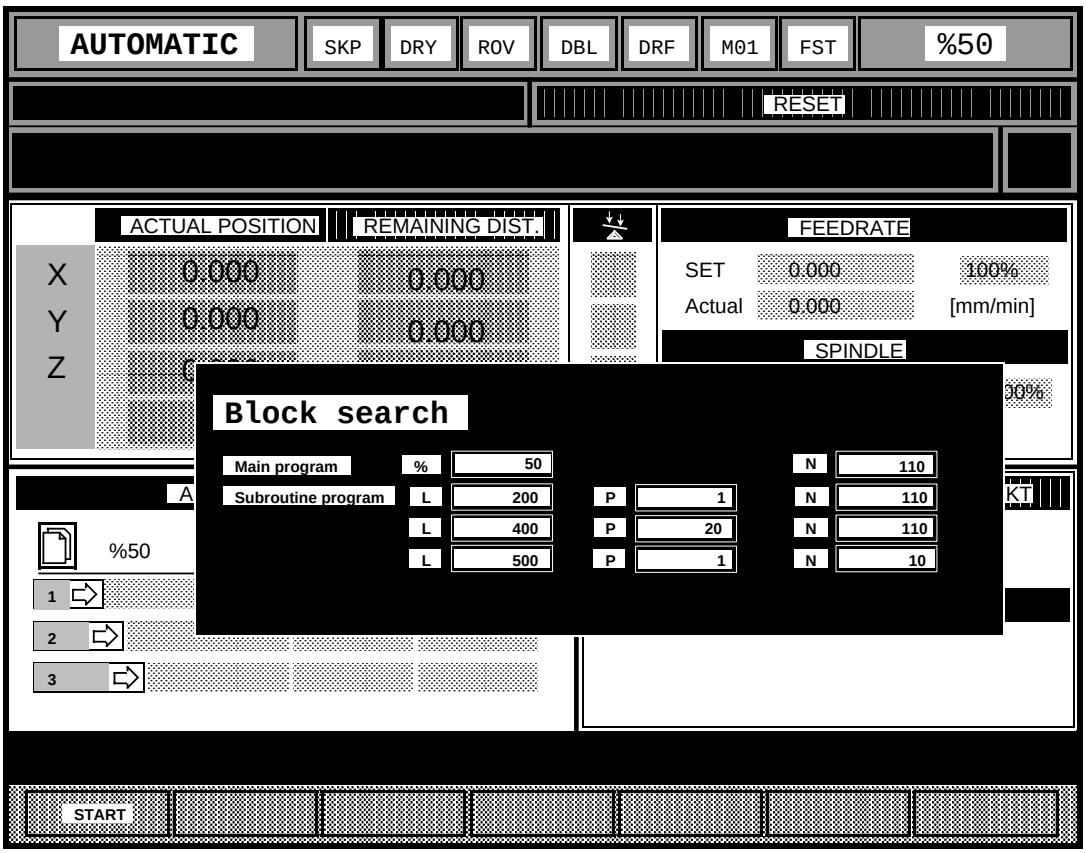
A block search in a nest of routines is likewise possible. For this purpose, enter in the main program the number of the block in which the subroutine call is programmed. After this, enter the number, the number of passes and the block number of the subroutine.

Sequence of operation

Prerequisites: The control is in the RESET state. The basic menu for the AUTOMATIC mode has been selected.

BLOCK SEARCH

The following window is opened by operating the BLOCK SEARCH key:

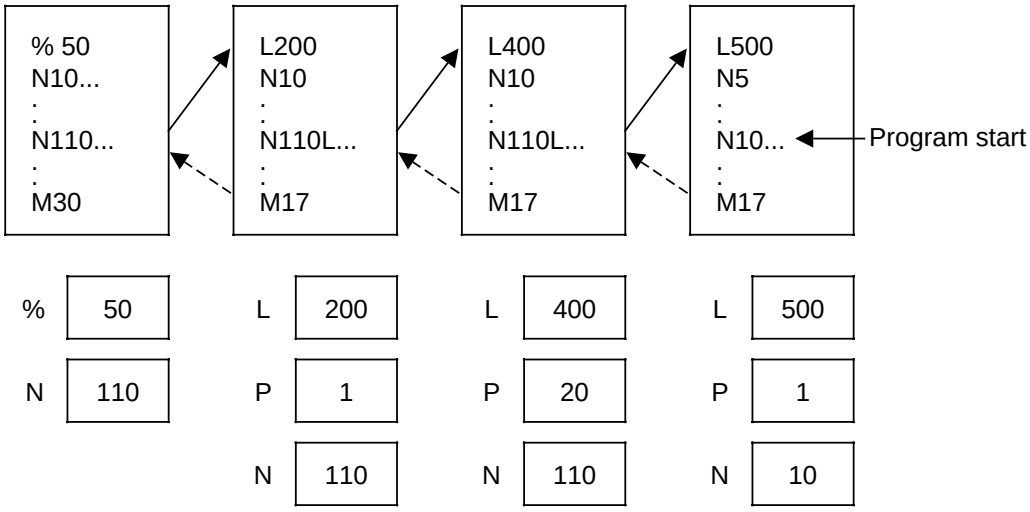


© Siemens AG 1990 All Rights Reserved

Now enter the following data required for the search in the appropriate input fields as described in Section 2.4.4 (Entry in Input fields).

L: Subroutine number
N: Block number
P: Search number

Example:



You can close the BLOCK SEARCH window by operating the RECALL key. The function is not carried out.

Operate the START key. The program is processed up to the search destination without any axis movement. The BLOCK SEARCH window is closed.

Operate the NC-START key to recommence processing of the program from the search destination onwards.

Note:

After a BLOCK SEARCH, you can traverse the difference between the actual position and the calculated set position by means of the axis selection and the direction keys in the JOG-REPOS mode.

Using the HOME key you can select the input field for the block number of the main program in the BLOCK SEARCH window irrespective of the selected input field.

© Siemens AG 1990 All Rights Reserved

5.3.5 PROGRAM CONTROL function

You can select this function by means of softkey in the AUTOMATIC and MDA modes. The following program controls can be activated or deactivated:

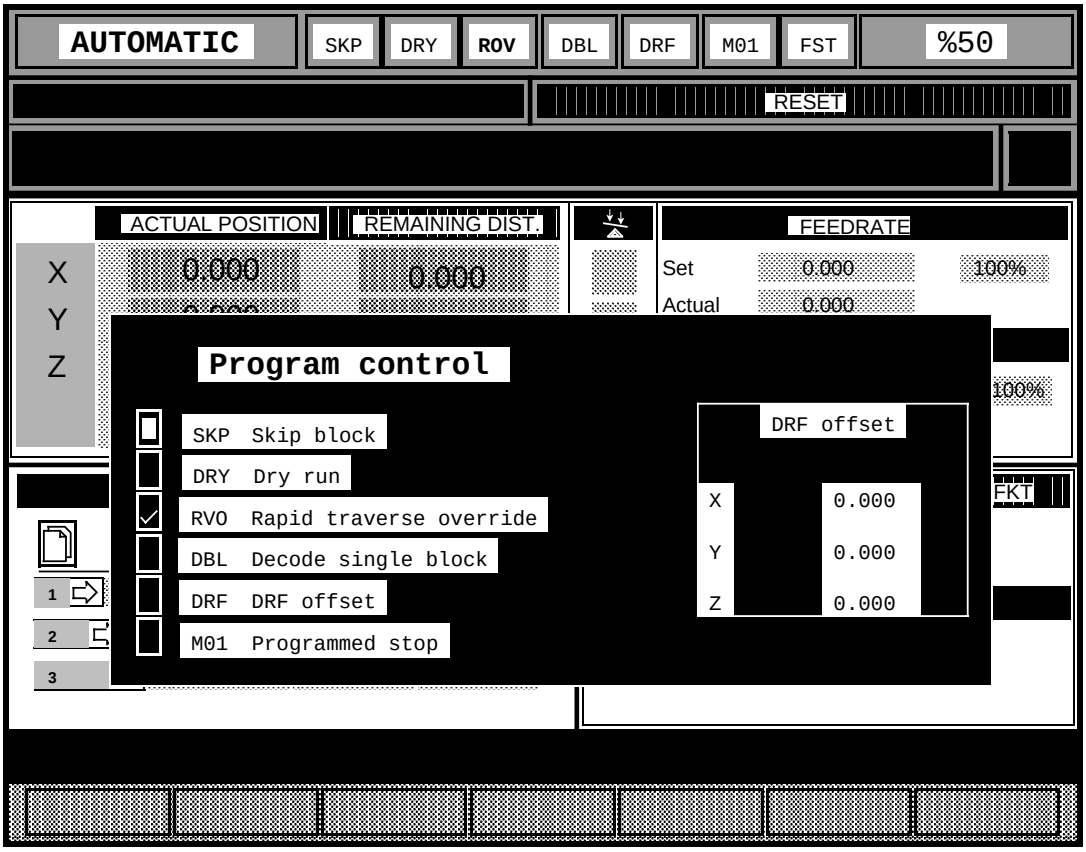
- SKP Skip block
- DRY Dry run
- ROV Rapid traverse override
- DBL Decode single block
- DRF DRF offset
- M01 Programmed stop

Sequence of operation

Prerequisite: You have selected the AUTOMATIC mode (i.e. MDA) in the machine area using the appropriate key.

Operate the softkey PROGRAM CONTROL to open the following window in the basic menu of the AUTOMATIC mode (i.e. MDA):

© Siemens AG 1990 All Rights Reserved



Use the cursor keys to move the white input field to the desired position.

Use the PLUS/MINUS keys to activate/deactivate the selected function. The input field indicates whether the function is active.

☐ ≙ function not active

☑ ≙ function active

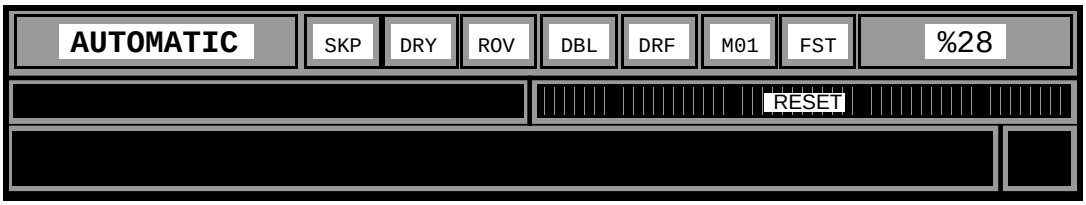
After you have activated/deactivated the functions as required, close the window using the RECALL key.

Note:

The program control status display marks the program controls which have been activated (in the example given: ROV).

© Siemens AG 1990 All Rights Reserved

Explanation of PROGRAM CONTROL basic menu



The status display is shown irrespective of the menu selected.

Program control	**SKP**	**skip block**

Program blocks which are marked with an oblique in front of the block number, are ignored during the processing of the program (e.g. "/100...").

Program control	**DRY**	**dry run**

Traversing is carried out at the feedrate value entered in the "dry run feedrate" setting data.

The programmed feedrate in program blocks G01, G02, G03, G33, G34 and G35 is no longer valid.

The dry run feedrate also applies instead of the programmed feed per revolution in program blocks with G95.

Program control	**ROV**	**rapid traverse override**

There are two possible ways of applying this function:

a) There is a common override switch for the feedrate and rapid traverse override.

ROV ☑ : The value of the feedrate override switch with an override value limit of 100 % is valid for the rapid traverse override.

ROV ☐ : Rapid traverse override is not possible.
The override value of 100 % is stipulated.

b) There is a separate override switch for the rapid traverse override.

ROV ☑ : The values set with the rapid traverse override switch are active.

ROV ☐ : The values set with the rapid traverse override switch are not active. The override value of 100 % is stipulated.

© Siemens AG 1990 All Rights Reserved

Example:

During installation of a new part program, the override switch also affects the traversing blocks with programmed rapid traverse (G00) when a function is activated.

Program control	**DBL**	**decode single block**

When the function is activated, the part program blocks are processed individually as follows:

Operate the NC-START key to process a decoding single block.

Operating the NC-START key again processes the next program block ...etc.

Notes:

- The SINGLE BLOCK function operates similarly.

Operate the SINGLE BLOCK key to activate the function.

SINGLE BLOCK SELECTED appears on the screen.

After the NC-START key has been operated, the program block is processed.

Operate the NC-START key again to process the next program block ...etc.

You deselect the function by operating the SINGLE BLOCK key again.

© Siemens AG 1990 All Rights Reserved

The following table indicates for 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	X	X
Blocks without path information	X	X
Calculation blocks		X
Switching and auxiliary function blocks	X	X
Internally generated control blocks (CRC...) **)	X	X
Threading blocks without dry run feedrate	*)	
Threading blocks with dry run feedrate	X	X

X "Single block" or "Decoding single block" signal is required.

*) Single block: The stop is made at the end of the first non-threading block

**) 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" (Cap. 5.3.12).

Program control	**DRF**	**DRF offset**

Using the "DRF" function (Differential Resolver Function) it is possible to set an additional, incremental zero offset using the handwheel.

This offset is not taken into account in the actual value display.

Notes:

- The active zero offset for each axis is only shown in the opened PROGRAM CONTROL window.
- The DRF offset for an axis cannot be deleted. The offset must be "moved back".
- The handwheel incremental dimension is defined as 1.
- See Section 5.3.7, DRF Function, for further information.

© Siemens AG 1990 All Rights Reserved

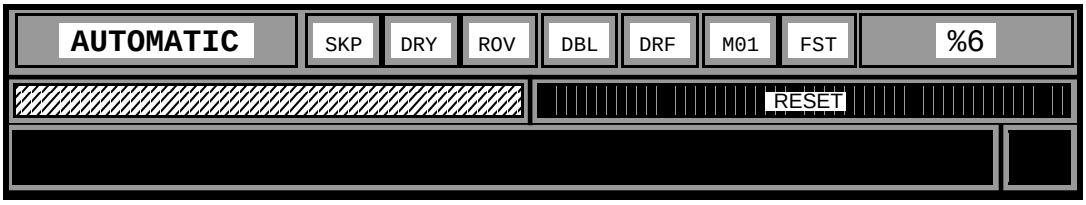
Program control	**M01**	**Programmed stop**

When this function is activated, the program is stopped at those blocks in which the additional function M01 is programmed.

STOP: PROGR.STOP M00/M01 is then displayed on the screen.
Processing of the program is restarted using the NC-START key. If the function is not active, the additional function M01 is not taken into consideration.

5.3.6 Program control in AUTOMATIC mode

Any program controls activated in the AUTOMATIC mode i.e. as a result of programmed or selected functions, are displayed in the **shaded display field**.



The following displays are possible:

A current program has been interrupted with the NC-STOP key. You can restart the program at the point of interruption using the NC-START key.

This display appears after the processing of each single block in the SINGLE BLOCK function.

An interruption has been programmed with M00 (progr. interruption unconditional) or M01 (progr. interruption conditional) in a part program. This display appears when the block is processed with M00 or, if the program control PROGRAMMED STOP has been activated, with M01. Processing of the program is recommenced by operating the NC-START key.

This display appears during the processing of an NC block with programmed dwell time.

© Siemens AG 1990 All Rights Reserved

Stop: Read-in enable

The interface signal READ-IN ENABLE has been removed. The current NC block is still being processed. Processing of the program with the next block is continued by setting the READ-IN ENABLE signal.

5.3.7 DRF function

The DRF function (Differential Resolver Function) makes an additional incremental zero offset using the handwheel possible.

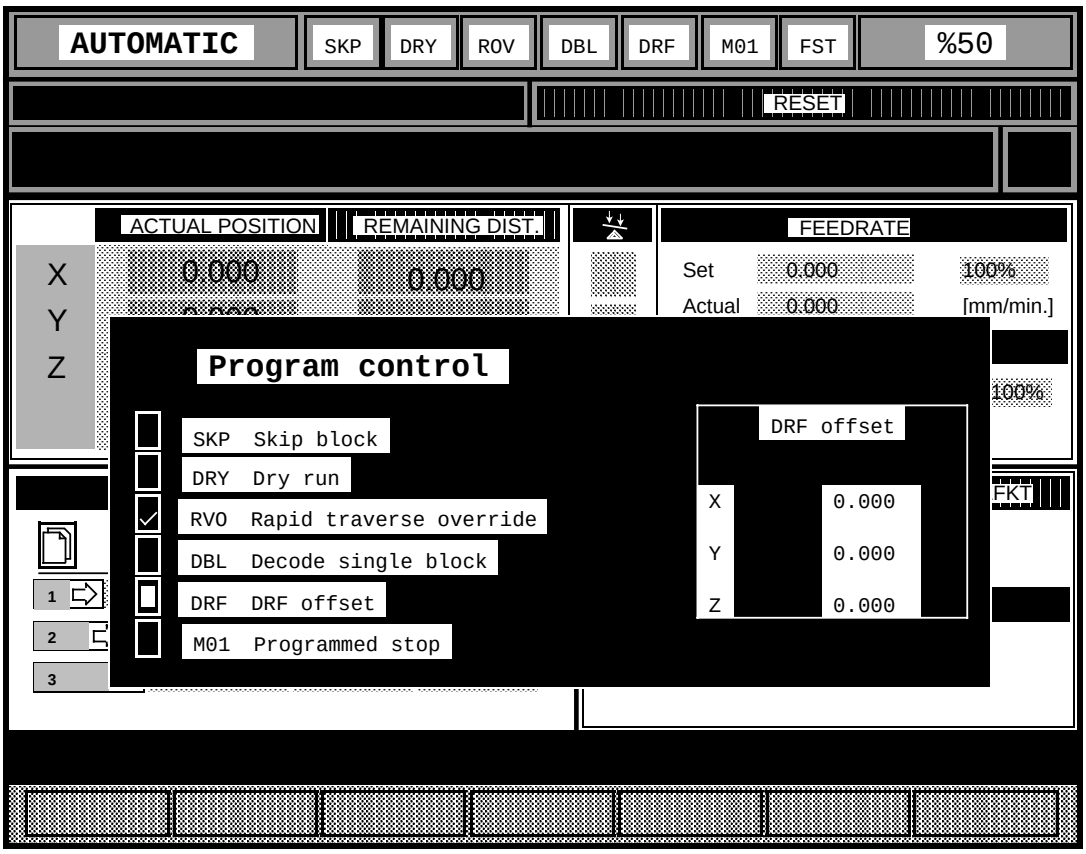
This zero offset is not taken into account in the actual value display.

Sequence of operation

Prerequisites: The MDA or AUTOMATIC mode basic menu has been selected.

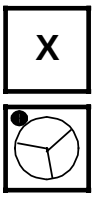
PROGRAM CONTROL

Operate the PROGRAM CONTROL softkey to open the following window:



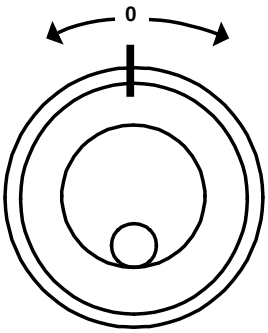
© Siemens AG 1990 All Rights Reserved

Now activate the DRF function as described in Section 5.3.5 (Program control).



Now select the axis to which the DRF offset is to be applied (e.g X axis).

Activate the HANDWHEEL function by operating the appropriate key on the handwheel.



Now turn the handwheel in the positive or negative direction according to the desired offset. The DRF offset value is displayed in the opened PROGRAM CONTROL window (X axis in the example).
The same procedure as above applies when altering the other axes.

Notes:

- The handwheel incremental dimension in the DRF function is always 1.
- It is not possible to delete the DRF offset of an axis. The axis must be "returned" by the amount of the offset.

© Siemens AG 1990 All Rights Reserved

5.3.8 Traversing in the JOG mode

In this mode, operation is carried out manually. You traverse the axes using the direction keys or using the handwheel.

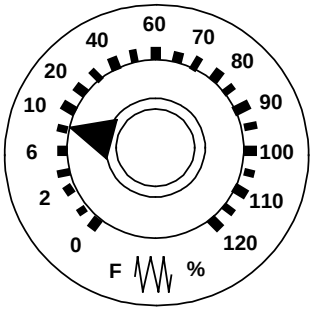
a) Traversing in JOG mode using the direction keys

Sequence of operation

To select the JOG mode operate the appropriate key on the operator keyboard or on the hand-held unit.

Now select the axis to be traversed by operating the appropriate selection key (e.g. X axis) on the handheld unit.

Use the JOG PLUS and/or MINUS keys to traverse the selected axis in the required direction.



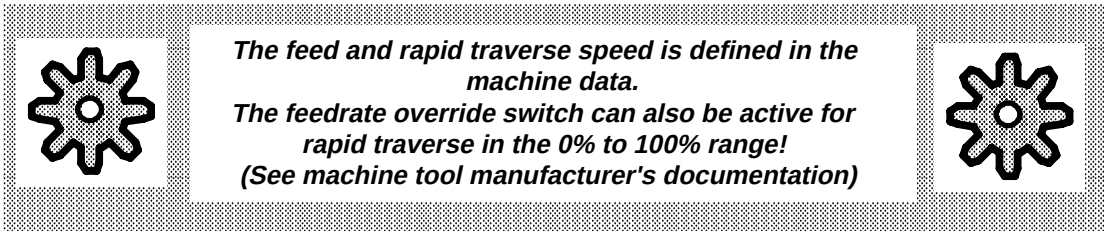
The traversing speed can be modified using the feedrate override switch.
The "0%" setting generally causes FEED STOP in the case of feed and rapid traverse.

The axis is traversed at rapid traverse rate by operating the JOG PLUS or MINUS key together with the RAPID TRAVERSE OVERRIDE key.

© Siemens AG 1990 All Rights Reserved

Notes:

- FEED STOP (FST) must not be on.
- The spindle speed as well as the T, H and M functions are determined by means of the OVERSTORE function.



b) Traversing in JOG mode using the handwheel

Sequence of operation

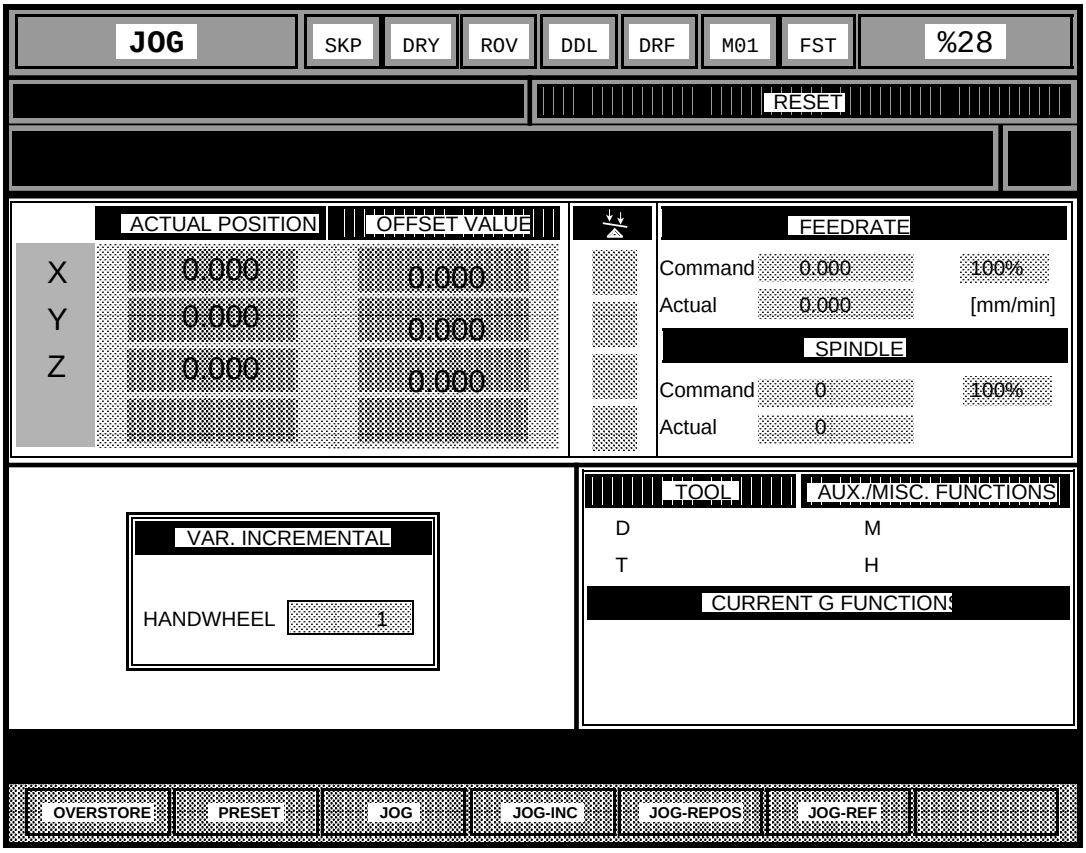
When the HANDWHEEL is active, the control traverses the selected axis by the increment selected in INCREMENTAL DIMENSION using the +1 and -1 keys (values 1, 10, 100) when 1 impulse is set by the handwheel.

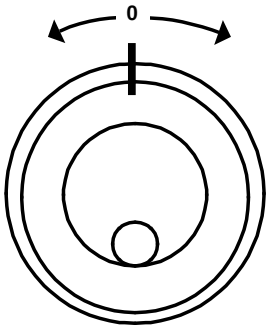
Select the JOG mode using the appropriate key on the operator keyboard or using the hand-held unit.

Use the axis selection keys to specify the axis which is to be traversed (e.g. Z axis).

Activate the HANDWHEEL function by operating the appropriate key on the hand-held unit. The corresponding LED lights up.

© Siemens AG 1990 All Rights Reserved





Turn the handwheel according to the desired traverse direction.
When 1 impulse has been set by the handwheel, the control traverses the selected axis by the increment entered in the INCREMENTAL DIMENSION HANDWHEEL entry field.

Notes:

- The values 1, 10 or 100 can be entered in the INCREMENTAL DIMENSION HANDWHEEL input field.
- Operating the HANDWHEEL key again deselects the function.
  The corresponding LED goes out.

© Siemens AG 1990 All Rights Reserved

5.3.9 Traversing in the JOG-INC mode

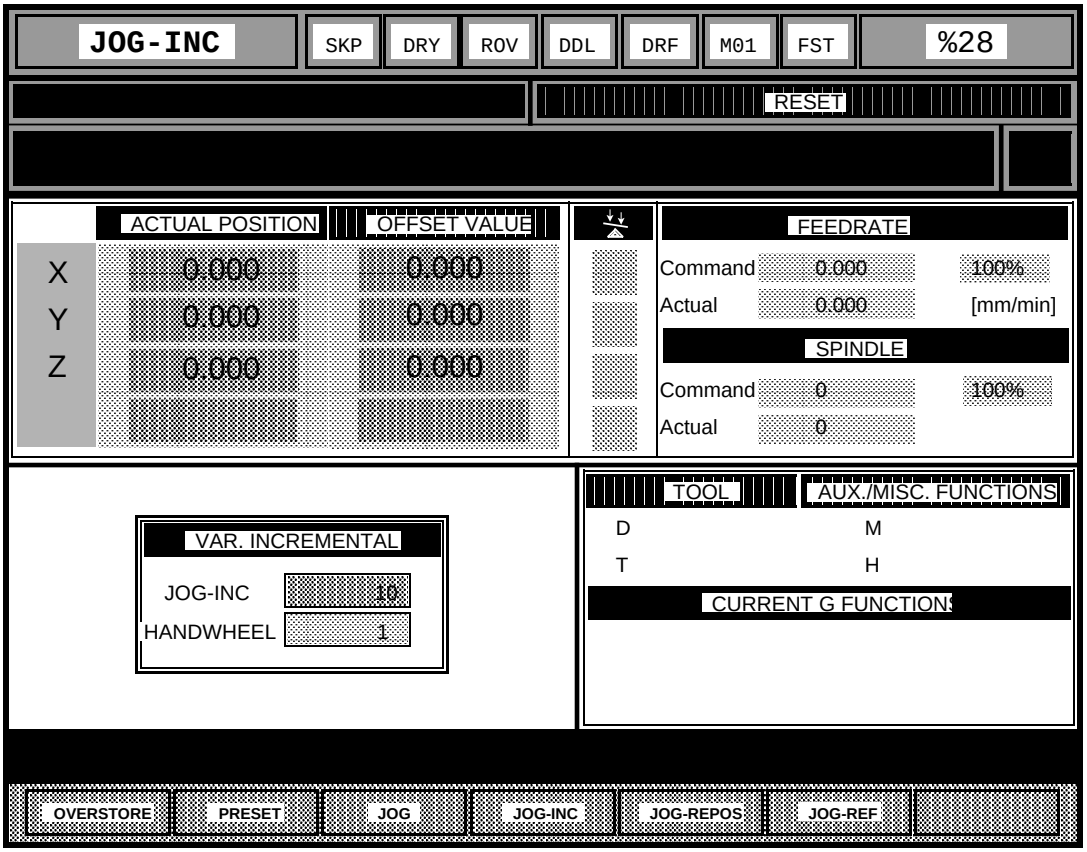
In this mode, operation is carried out manually via the control. With each operation of a direction key, you traverse the axis concerned in the selected direction by the set increment. In this mode, traversing can also be carried out using the handwheel.

a) Traversing in the JOG-INC mode using the direction keys

Sequence of operation

To select the INC mode operate the appropriate keys on the operator keyboard or on the hand-held unit.

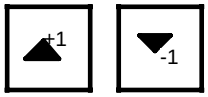
Now operate the JOG-INC softkey.



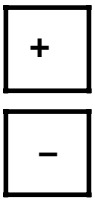
Using the axis selection keys, specify the axis to be traversed.

© Siemens AG 1990 All Rights Reserved

Now enter the desired increment (between 1 and 10000) in the JOG-IN INCREMENTAL DIMENSION input field, as described in Section 2.4.4 (Entry in input fields).



By operating the increment/decrement key you can switch through the increment amount in the JOG-INC field in the INCREMENTAL DIMENSION window.
Possible values: 1, 10, 1000, 10000



Operate a direction key to traverse the selected axis by the increment set by you.

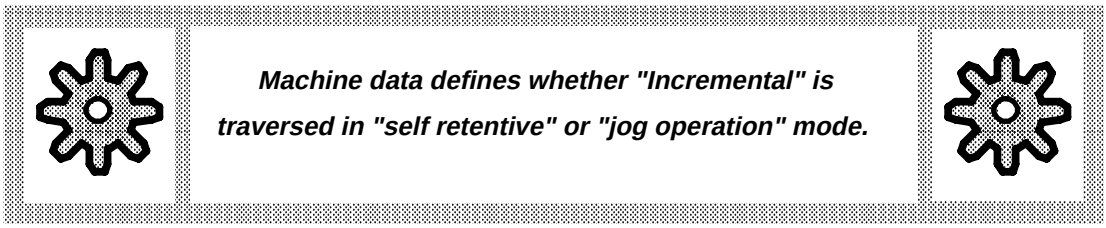
Note:

The direction keys can have two different characteristics:

- **"Self retentive (modal)"**
- **"JOG operation"**

With **"self retentive"** the axis is always traversed by one increment, depending on the setting , 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.



© Siemens AG 1990 All Rights Reserved

b) Traversing in the JOG-INC mode using the handwheel

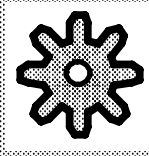
To select the JOG mode use the appropriate key on the operator keyboard or on the hand-held unit.

Now operate the JOG-INC softkey.

Sequence of operation

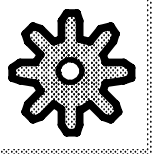
Notes:

- The continued sequence of operation is described in Section 5.3.8 (Traversing in the JOG mode with the handwheel), in b).
- FEEDRATE STOP must not be on.
- Values between 0 and 10 000 can be entered in the input field for JOG-INC incremental dimension.
- The spindle speed as well as the T, H and M functions are specified using the OVERSTORE function.



The feedrate is defined as machine data!
The machine data also defines whether JOG-IN is traversed in "self retentive" or "jog operation" mode!

(See machine tool manufacturer's documentation.)



© Siemens AG 1990 All Rights Reserved

5.3.10 Traversing in JOG-REPOS mode

After a program interruption - e.g. after switching from AUTOMATIC operation to JOG or INCREMENTAL DIMENSION, 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.

A RESET is not generated when switching from AUTOMATIC mode to:

- JOG or to
- JOG-INC or to
- JOG-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 "Repositioning" JOG-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.

After a tool change, you can only use the mode with the same tool dimensions as before.

Otherwise you should use:

- the "block search" method (s. Section 5.3.4).
- the "scratch" method (s. Section 5.3.16).

© Siemens AG 1990 All Rights Reserved

Sequence of operation

Prerequisite: You have interrupted the program - e.g. by operating the NC-STOP key - and subsequently have traversed away from the contour in the JOG mode.

JOG-REPOS

Operate the JOG-REPOS softkey (selects the JOG-REPOS mode).

X

Select the axis which is to be traversed to the point of interruption.

+ or **-**

Operate the direction key to traverse to the point of interruption. Overtravel past the point of interruption is inhibited by the NC. The feedrate override switch is active, the rapid traverse override is inactive.

Note:

The direction keys can have two 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 µm), 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.

Machine data defines whether "Incremental" is traversed in "self retentive" of "jog operation" mode.

Once all axes have been traversed to the point of interruption, select the AUTOMATIC mode. Operate the NC-START key to continue the program.

© Siemens AG 1990 All Rights Reserved

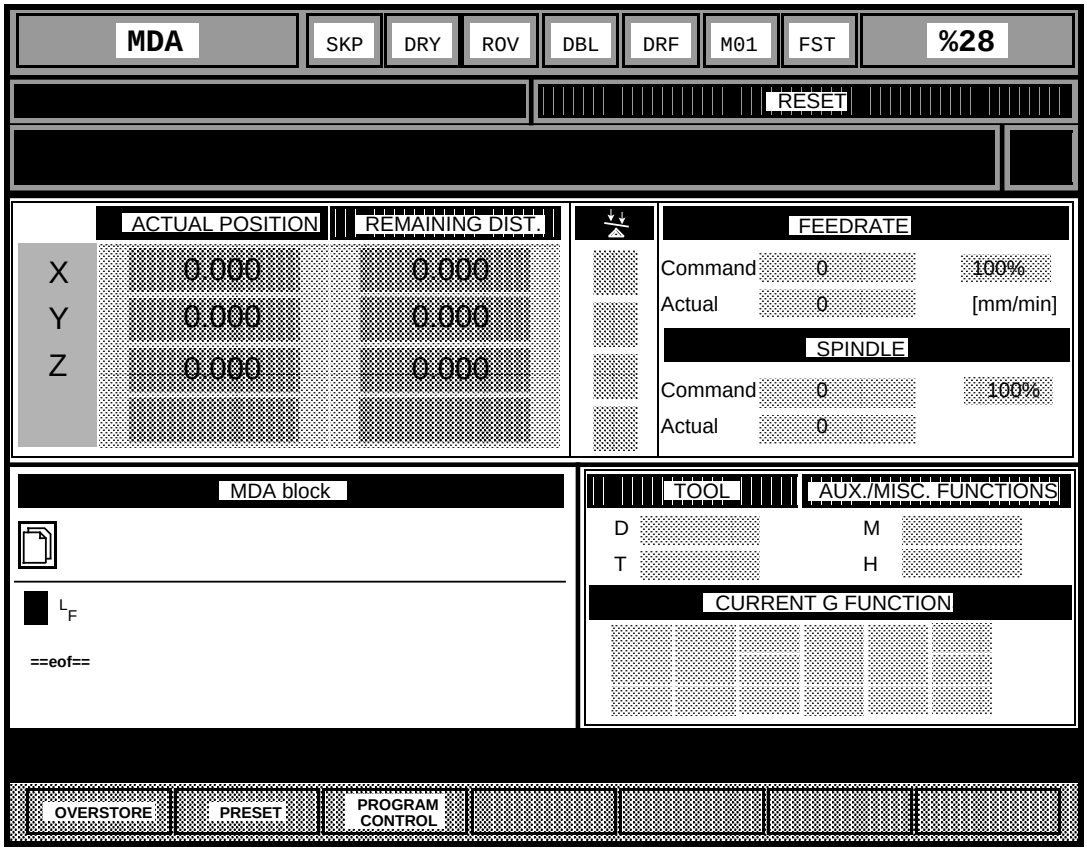
5.3.11 MDA mode

Single NC blocks can be entered and processed in this mode. After processing the single block is deleted.

Sequence of operation

Prerequisite: The SINUMERIK 805 machine area has been selected.

Operate this key to select MDA mode. the following display appears:



© Siemens AG 1990 All Rights Reserved

…

…

…

Using the keyboard enter the block according to the instructions in the Programming Guide and terminate each entry using the input key.

After operating the NC-START key the entered block is processed by the control and subsequently deleted.

Note:

You can process several blocks one after the other. The "retentive input data" (e.g. feedrate) are retained.

They are deleted or reset by:

- changing the operating mode or
- RESET

© Siemens AG 1990 All Rights Reserved

5.3.11.1 OPERATOR PROMPTING in MDA mode

The OPERATOR PROMPTING function is to activate via NC-MD 5150.2.

In MDA mode you can call a subroutine in a block. If this subroutine is fitted out with operator prompting (see Programming Guide Section 12), this function supports a comfortable parameter assignment.

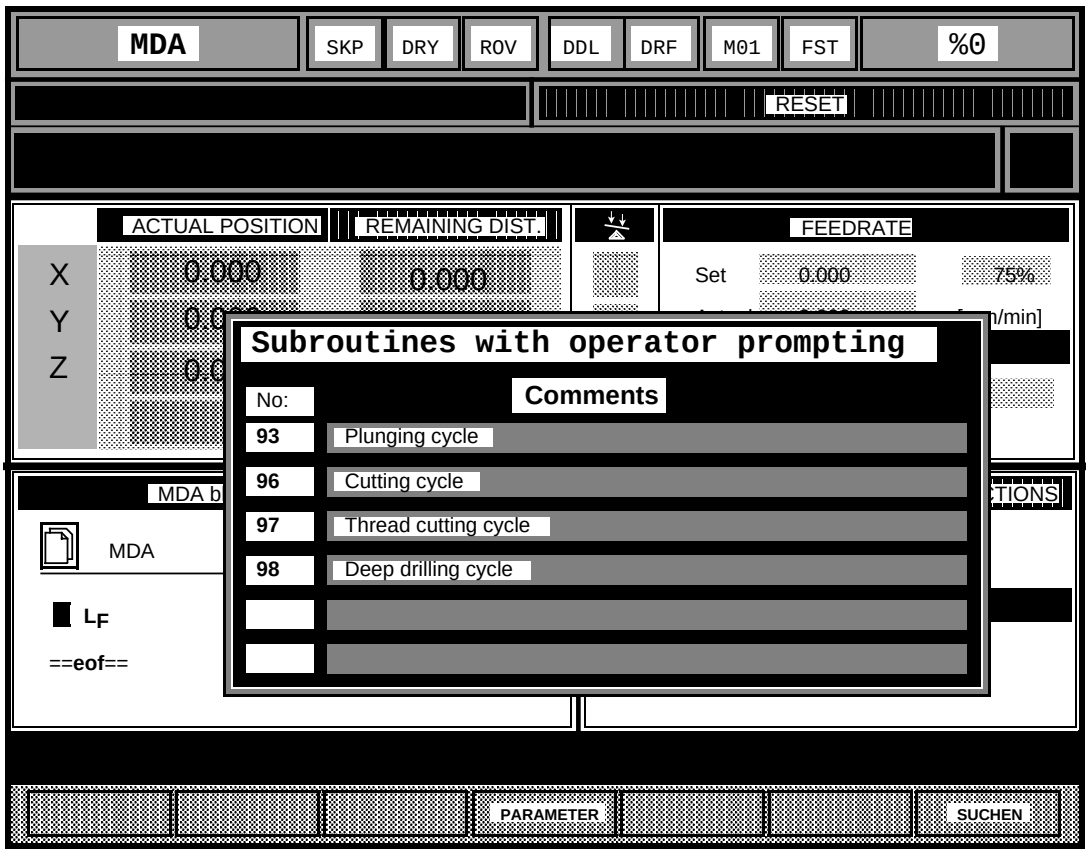
a) Manual subroutine selection

Sequence of operation

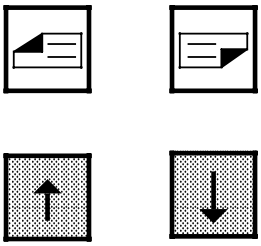
Prerequisite:: MDA mode has been selected.

OPERATOR PROMPTING

Operation of OPERATOR PROMPTING softkey opens a window listing all subroutines fitted out with operator prompting.



© Siemens AG 1990 All Rights Reserved



Exist more than 6 subroutines with operator prompting they can be displayed by operating PAGE UP/ DOWN softkeys.

Using the cursor keys you can select the desired subroutine in the overview display (inverse displayed ≙ selected).

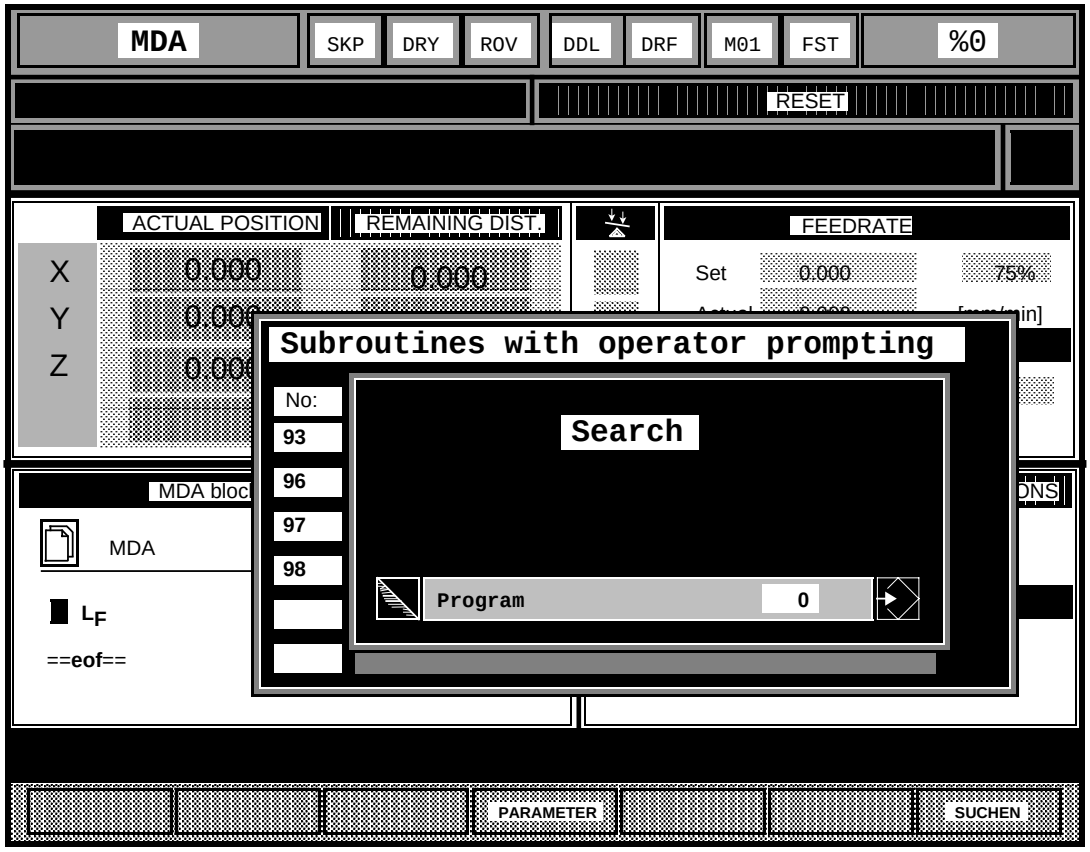
b) Subroutine selection using the SEARCH softkey

Sequence of operation

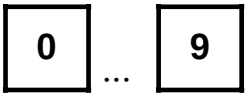
Prerequisite: MDA mode has been selected.

Operating of OPERATOR PROMPTING softkey opens a window listing all subroutines fitted out with operator prompting.

Press the SEARCH softkey and the following window appears:



© Siemens AG 1990 All Rights Reserved

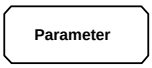


Key in the desired subroutine number in the opened entry field.

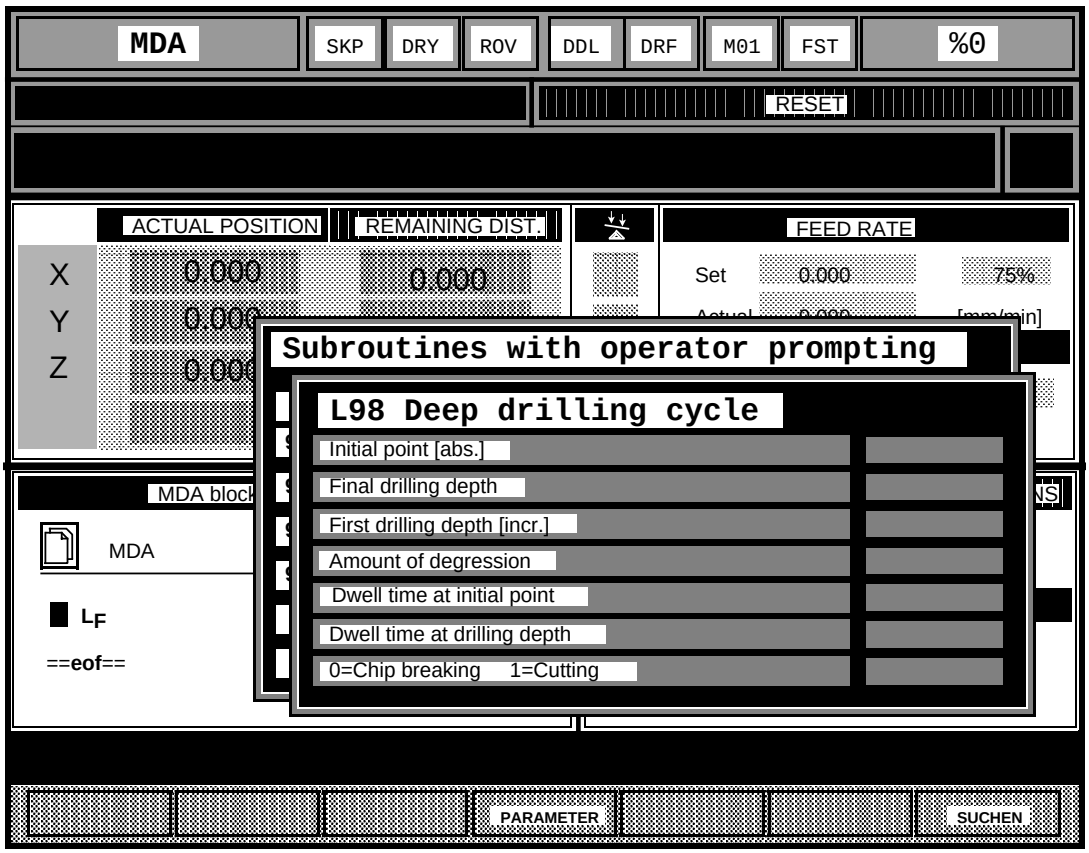
Operation of INPUT key closes entry field and window.
Now the wanted subroutine is selected (inverse displayed).

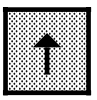
Parameter assignment for the selected subroutine

Prerequisite: The desired subroutine has been selected.



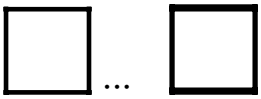
Operation of PARAMETER softkey opens another window listing all subroutines fitted out with operator prompting.



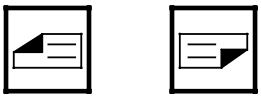


Use cursor keys for R parameter selection.

© Siemens AG 1990 All Rights Reserved



Now entry the parameters after description in Section 2.4.4 (Entries in Input fields).



Exist more than 7 parameters for the subroutine they can be displayed by operating PAGE UP/DOWN softkeys.

After operating the ACCEPT DATA softkey the part program accepts the entried R parameter assignments.
After all R parameter assignments the subroutine call automatically is entried.
The PARAMETER and DIRECTORY windows are closed.

Note:

In case of repeated successive subroutine calls in MDA mode the assignment of all parameters is only necessary at the first call.
At later subroutine calls key in the parameters only if their values have changed.

© Siemens AG 1990 All Rights Reserved

5.3.12 OVERSTORE function

In modes JOG, MDA and AUTOMATIC, you can modify one or more values of the following functions in the buffer memory:

- Auxiliary function H
- Miscellaneous function M
- Spindle speed S
- Tool number T

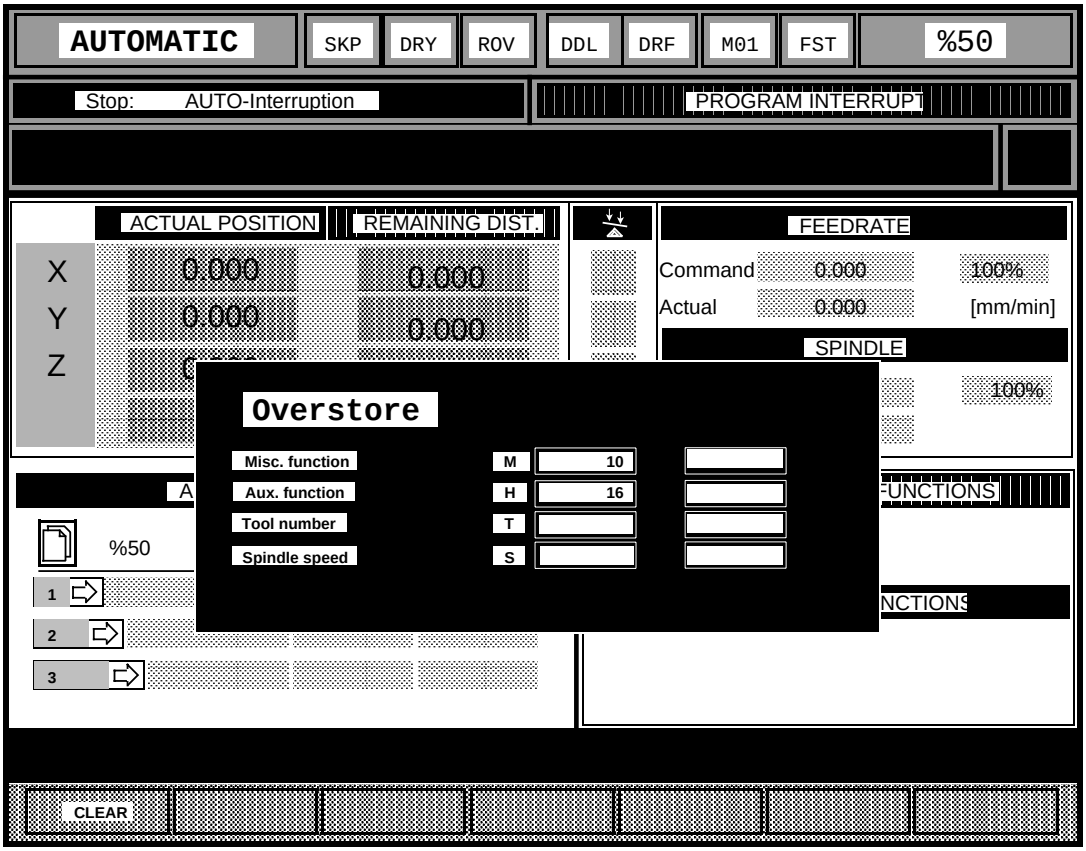
Sequence of operation for entering values

Prerequisites:

- The basic menu of the above mentioned modes is selected.
- To OVERSTORE, the program must be stopped using the NC-STOP key.

OVERSTORE

The following window is opened when you operate the OVERSTORE softkey:



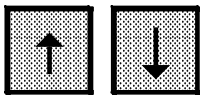
© Siemens AG 1990 All Rights Reserved

The active functions are entered in the fields on the left. In the fields on the right enter the required values as described in Section 2.4.4 (Entry in input fields).

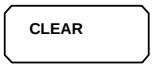
When the NC-START key is operated, the entered values become active and the input fields are deleted. The active values can be modifed using the OVERSTORE function.

Operate the RECALL key to close the window.

Sequence of operation for deleting an entered value



Use the cursor keys to select the field of which the contents are to be deleted.



Operate the CLEAR key.
The entered value is deleted.

Notes:

- You can close the window with the RECALL key before operating the NC-START key. The OVERSTORE function is then not executed.
- In AUTOMATIC mode the program is now continued by operating the NC-START button with the new values.

© Siemens AG 1990 All Rights Reserved

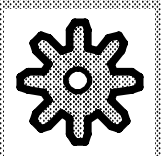
5.3.13 PRESET function (set actual value)

In the basic menu of modes, JOG, MDA and AUTOMATIC, you can select the PRESET function with the appropriate softkey.

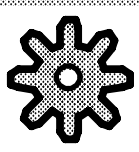
- In the ”PRESET” operating mode, you can offset the control zero point to any point within the machine coordinate system.
  You enter the value for the offset in the actual value memory (preset). The actual value memory for all available axes can be preset. This preset results in a ”PRESET offset” which is displayed on the screen.

- **No movement** of the axes takes place with ”Set actual value”. A new value is simply entered for the current axis positions.

- The ”PRESET offset” remains stored:
  - after end of program
  - after RESET



Machine data defines whether the PRESET offset is automatically deleted:
- on switching on the control
- on approaching reference point
(see machine tool manufacturer's documentation)

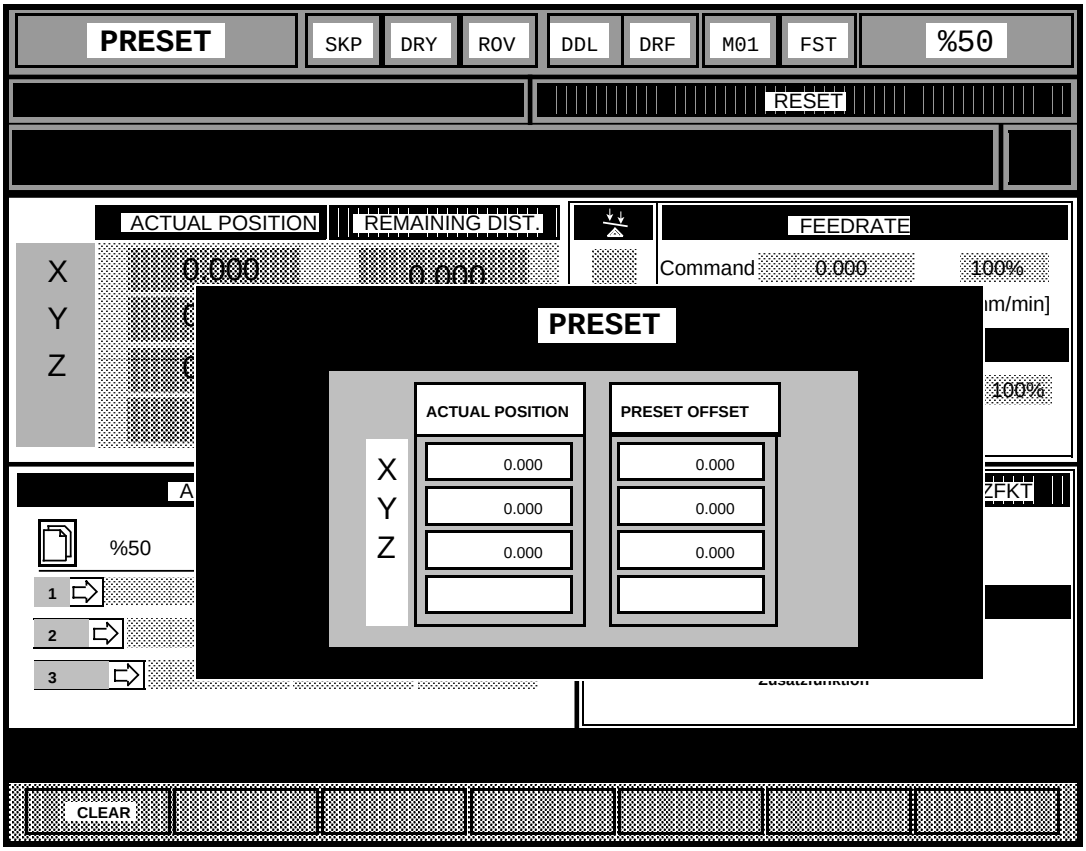


© Siemens AG 1990 All Rights Reserved

Sequence of operation when entering a PRESET value

Prerequisite: One of the following modes has been selected in the machine area: JOG, MDA, AUTOMATIC. The control is in the RESET state.

Operate the PRESET key to open the following window:



Now enter for each of the axes as described in Section 2.4.4 (Entry in input fields) the actual value which is to correspond to the present axis position in the future

After operating the RECALL button the window is closed and the actual value in the display altered to the new value.

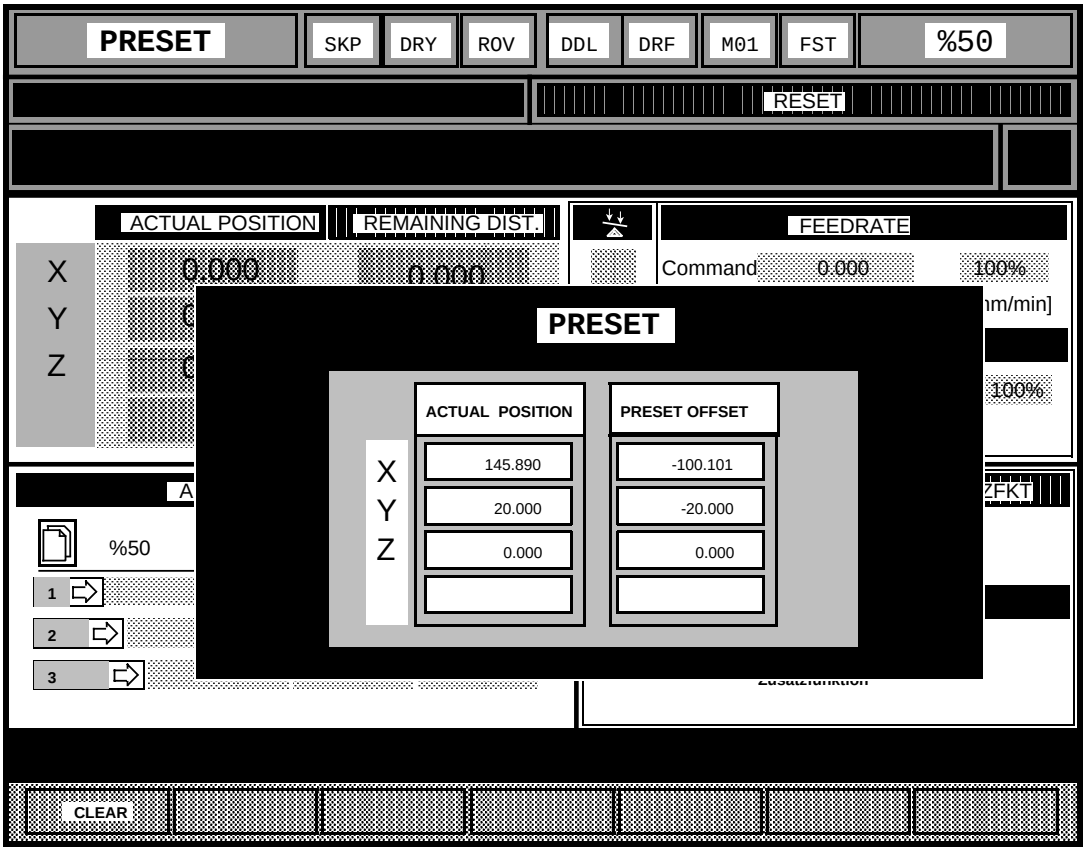
© Siemens AG 1990 All Rights Reserved

Sequence of operation when entering a PRESET value

Prerequisite: One of the following modes has been selected in the machine area: JOG, MDA, AUTOMATIC. The control is in the RESET state.

PRESET

Operate the PRESET key to open the following window:



Using the cursor keys select the axis for which the PRESET value is to be deleted.

Clear

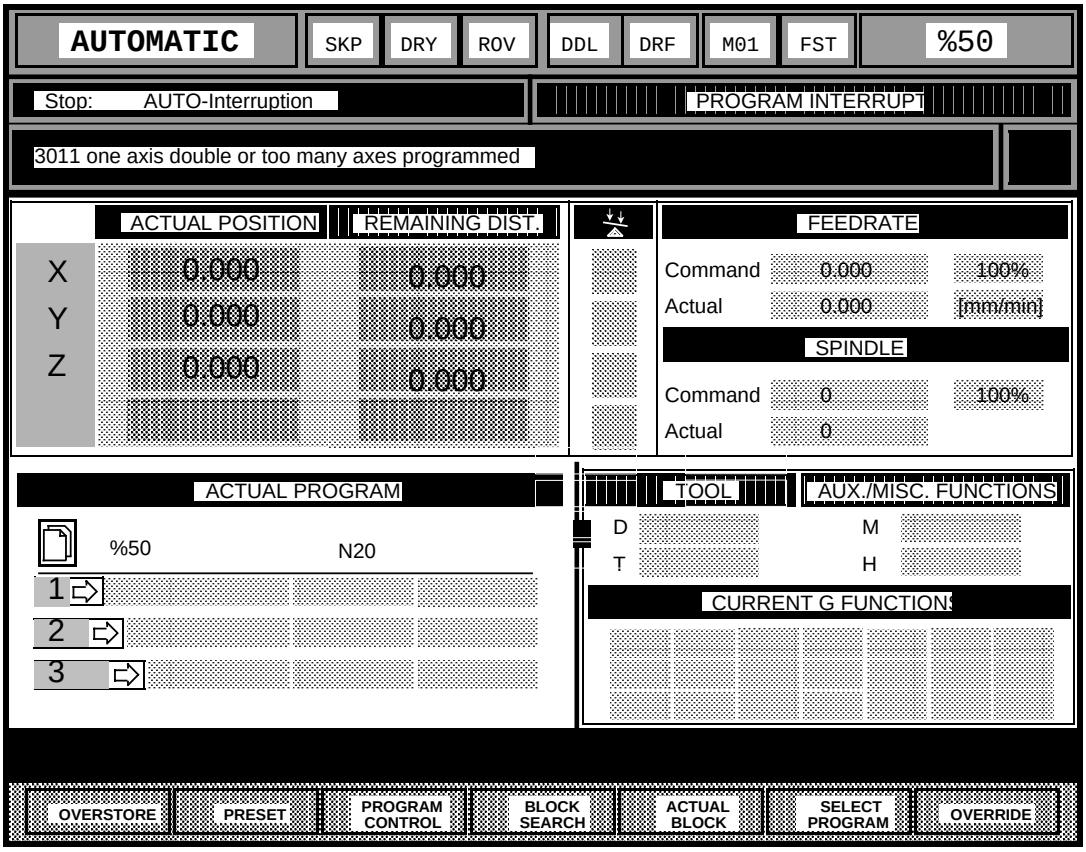
Now operate the CLEAR softkey. The PRESET OFFSET value is now deleted.

After operating the RECALL key the actual value display again refers to the machine coordinate system.

© Siemens AG 1990 All Rights Reserved

5.3.14 OVERRIDE BLOCK function

If the control recognizes a programming fault during program run the operation is stopped and a corresponding alarm displayed.
Additionally, the softkey line and the OVERRIDE BLOCK softkey are enlarged.

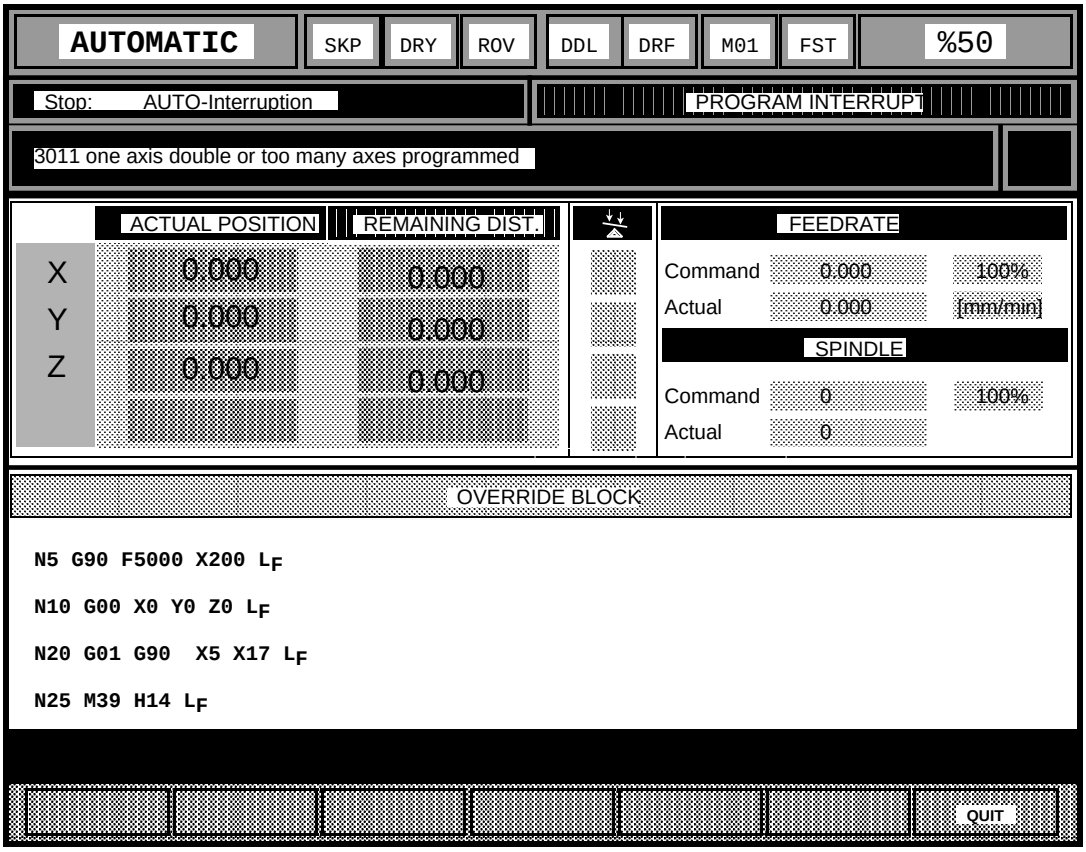


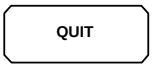
Operating the NC stop softkey interrupts the program.

Pressing the OVERRIDE BLOCK softkey opens a window where the cursor position is in the faulty program block.

Now you can change the faulty program block according to Section 4.3.1 ”Program editing”.

© Siemens AG 1990 All Rights Reserved





After correction of the faulty block the QUIT softkey **must** be operated to delete the current alarm.

Operating NC START softkey closes the window and continuates the program run with the corrected program part.

© Siemens AG 1990 All Rights Reserved

5.3.15 SPEED OVERLAY VIA HANDWHEEL PULSES function

SPEED OVERLAY VIA HANDWHEEL PULSES function is an optional function.

This function generates a speed overlay for an individual programmed axis in a block in AUTOMATIC and MDA mode.
Turning the handwheel in the positive or negative direction increases the programmed speed of the NC axis. The higher the handwheel speed resp. the incrementeal dimension of the handwheel are (Section 3.2.2) the higher is the speed overlay.

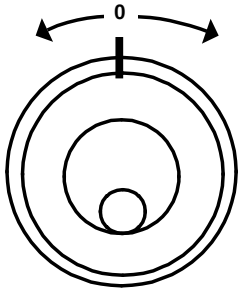
"DRF" function (Section 5.3.7), "dry run feedrate" (Section 5.3.5) and preparatory must be inactive.

Sequence of operation

Prerequisite: One individual NC axis has been programmed in the actual block.

Operate the axis selection key also programmed in the actual block (in this case X axis).

Operating the handwheel key activates the handwheel.



Turning the handwheel can preset a special speed overlay for the programmed axis.

© Siemens AG 1990 All Rights Reserved

5.3.16 Scratch

If a tool breaks during program processing, you must leave the contour by changing to the ”JOG” or ”JOG INC” 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:

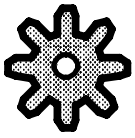
A tool change has been carried out and the approporiate correction data entered.

Using the direction keys, now traverse to any point within the interrupted block.
The axes can be traversed in the JOG or JOG-INC mode:

- using the direction keys
- using the handwheel

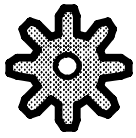
Select the AUTOMATIC mode using the appropriate key on the operator keyboard or the hand-held unit.

Resume program processing by operating the NC-START key on the hand-held unit.



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"; see also the machine tool manufacturer's documentation.)

If the range is exceeded: alarm 3018 !



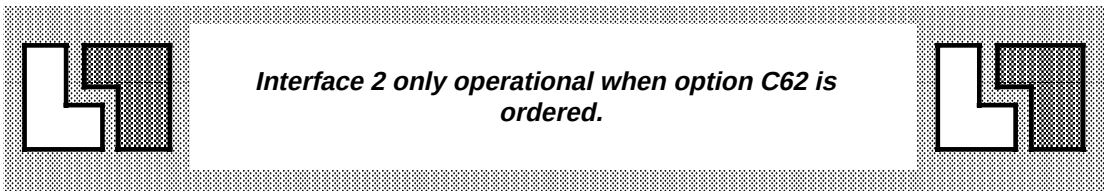
© Siemens AG 1990 All Rights Reserved

6 Data Interfaces

6.1 General

Two interfaces are available for data transfer with the SINUMERIK 805.

- Interface 1 (RS 232 C/20mA) is variable with regard to the possibilities for connecting different devices.
- Interface 2 (RS 232 C) is normally set aside for the connection of a particular device.



Notes:

- Interface 2 can also be operated with an RS 422 adaptor.
- The interface parameters (interface characteristics) for interfaces 1 and 2 can be set in the PARAMETER menu (See Section 4.8).

The following data types can be read-in/read-out via RS 232 C interfaces:

Data type	Meaning
CLF	Delete statement (**Cl**ear **F**ile), during read-in
MPF	Part program (**M**ain **P**rogram **F**ile)
RPA	R parameter with value assignment (**R-P**arameter **A**ctive)
SEA	Addresses with value assignments (**Se**tting Data **A**ctive)
SPF	Subroutine (**S**ub **P**rogram **F**ile)
TEA 1	NC machine data (**Te**sting Data **A**ctive)
TEA 2	PLC machine data (**Te**sting Data **A**ctive)
TOA	Tool offsets (**T**ool **O**ffset **A**ctive)
ZOA	Zero point offset (**Z**ero **O**ffset **A**ctive)

© Siemens AG 1990 All Rights Reserved

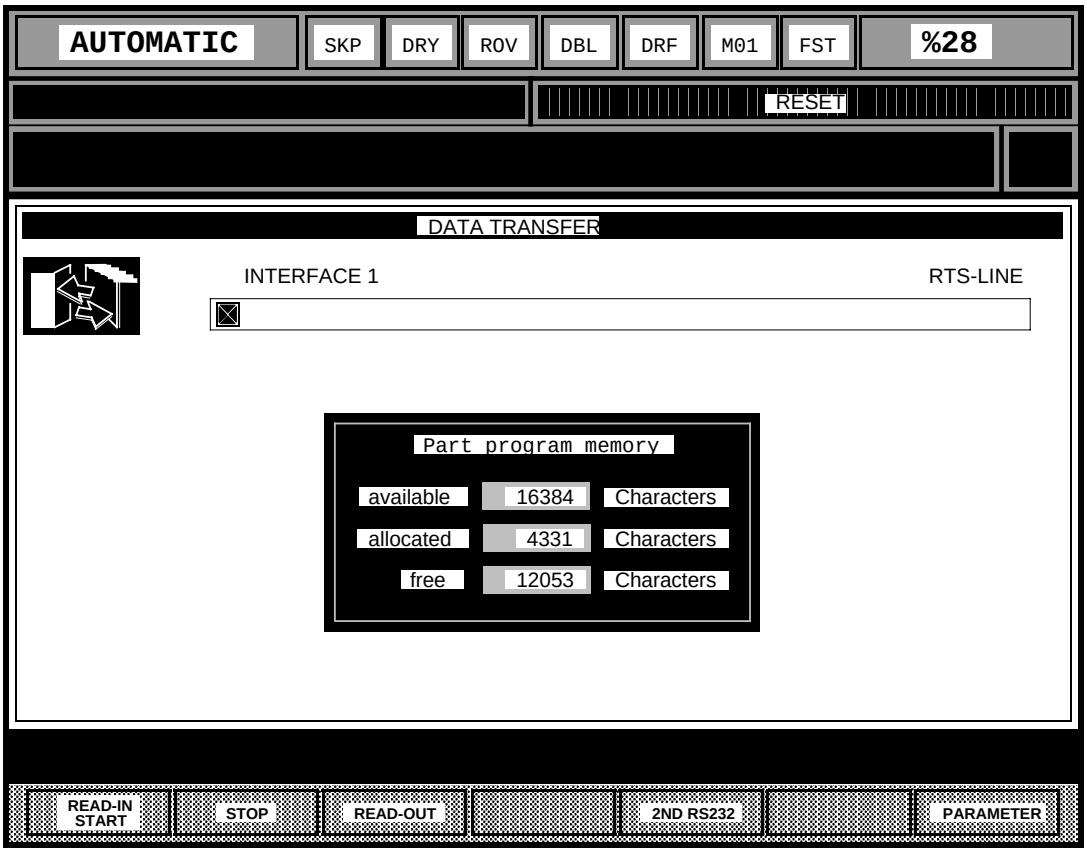
6.2 Data read-in via RS 232 C interface

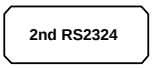
Sequence of operation

Prerequisite: The SINUMERIK 805 machine area has been selected.

Select the SINUMERIK 805 machine area using the OPERATING AREA key.

Operate the DATA TRANSFER softkey. The following screen display appears:





Select the desired interface by operating the appropriate softkey.

Interface 2 only operational when option C62 is ordered.

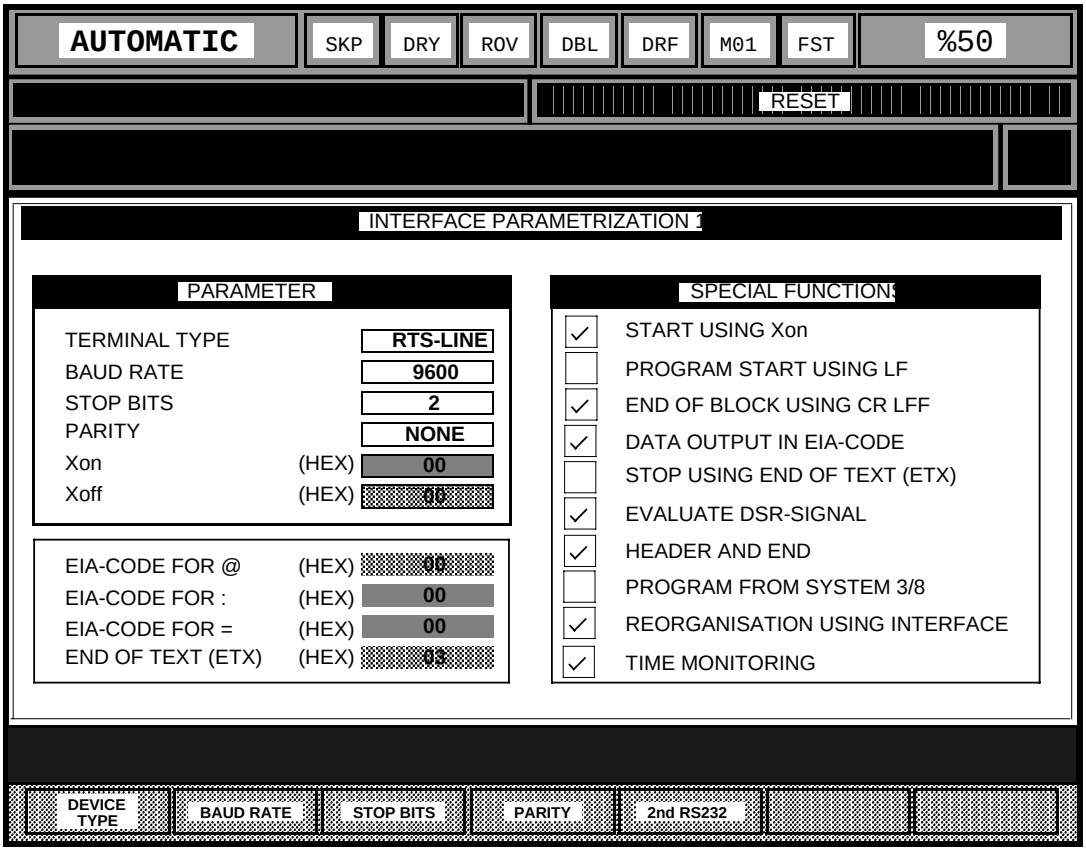
© Siemens AG 1990 All Rights Reserved

Sequence of operation when setting interface parameters

Prerequisite: The basic menu DATA TRANSFER has been selected.

PARAMETER

Operate the PARAMETER softkey to set or control the interface parameters. The following display appears:



© Siemens AG 1990 All Rights Reserved

Each time one of the four softkeys is operated, the corresponding setting is changed. The **following settings** are possible:

DEVICE TYPE	Device type: PLC-PROG, RTS-LINE, Xon/Xoff, PTR und PD/PF
BAUD RATE	Baud rate: 110, 150, 300, 600, 1200, 2400, 4800 and 9600
STOP BITS	Stop bits: 1/1, 5/2
PARITY	Parity: none, odd, even
↑	To input characters for Xon, Xoff, EIA code for @,: and = as well as for transfer end, select the corresponding input field using the cursor keys.
↓	Now enter the desired value as described in Section 2.4.4 (Entry in input fields).
→	To alter any of the special functions, select the appropriate input field using the cursor keys.
↑	
↓	
+/-	The special function is activated/deactivated each time the PLUS/MINUS key is operated. The input field indicates whether the function is active or not.

☐ =^ not active

☑ =^ active

© Siemens AG 1990 All Rights Reserved

Operate this key to return to the DATA TRANSFER basic menu.

READ-IN START

When this key has been operated, the control is ready to receive. The display "data transfer active" appears on the screen.

The data type sent by the transmitter (tape reader, programming unit) is characterized by its initial code.
The data type is displayed (e.g. TOA) during read-in.

STOP

Data input can be interrupted at any time using the STOP softkey. READ-IN start is continued when the softkey is operated again.

Please note:
The retrieval of specific data via the control is not possible when data is being input!

Notes:

In the case of tape input, the data input is checked for simple errors. All characters in a tape code have a common identifier.

- The number of punched holes in an ISO code is always even.
- The number of punched holes in an EIA code is always odd.

The control automatically recognizes the correct code as soon as the first "%" (ISO) or ”EOR” (EIA) has been read!
The criterion "number of punched holes even" or "number of punched holes odd" is used for a character parity check as from the second character.
As a further check, a program being read-in which is already contained in the program memory is compared thoroughly with its counterpart.
If an error is detected, the read-in process is halted and the error displayed on the screen.

© Siemens AG 1990 All Rights Reserved

6.3 Data read-out via RS 232 C interface

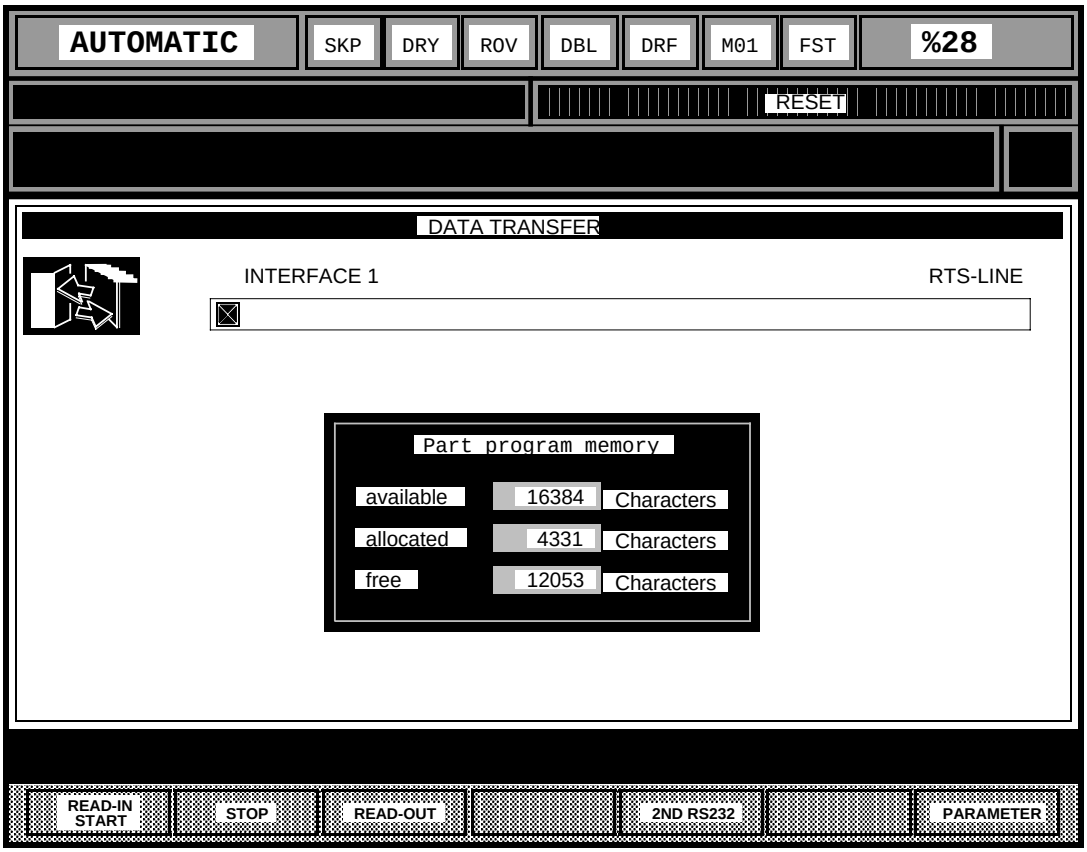
Operating sequence

Prerequisite: The SINUMERIK 805 machine area has been selected.

Use the OPERATING AREA key to select the SINUMERIK 805 data area.

DATA TRANSFER

The following display appears when you operate the DATA TRANSFER softkey:



2ND RS232

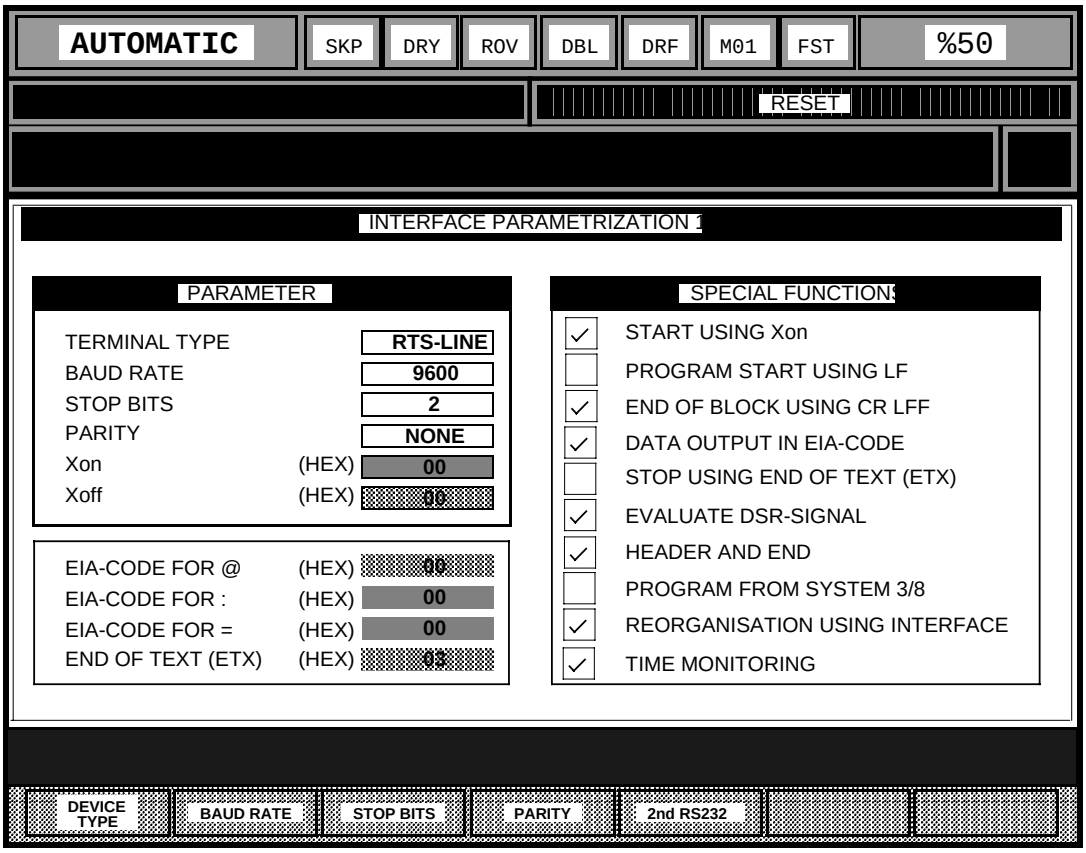
Select the desired interface by operating the appropriate softkey.

Interface 2 only operational when option C62 is ordered.

© Siemens AG 1990 All Rights Reserved

PARAMETER

Operate the PARAMETER softkey to set or control the interface parameters. The following display appears:



Notes:

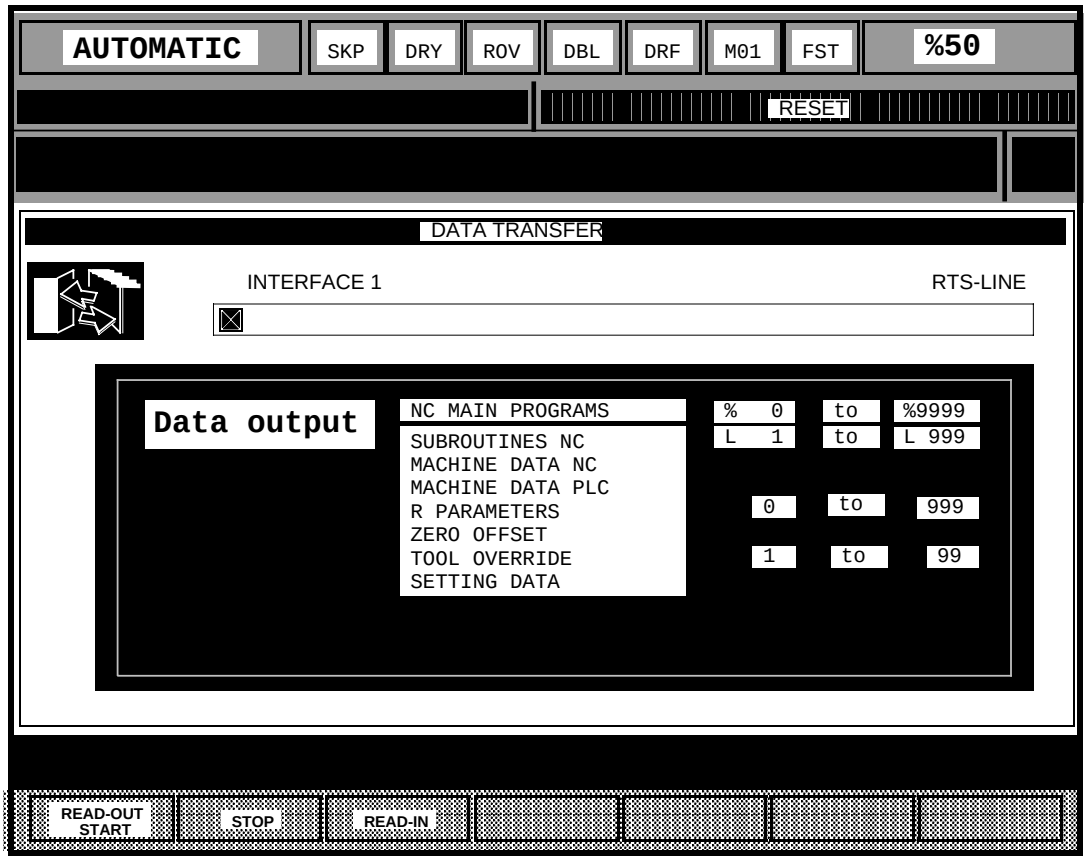
- The setting of interface parameters for various peripheral devices is described in Section 6.5.
- The input and modification of interface parameters is described in Section 6.2, "Data read-in via RS 232 C interface".

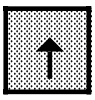
After you have entered the interface parameters, operate this key to return to the DATA TRANSFER basic menu.

© Siemens AG 1990 All Rights Reserved

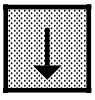
READ-OUT

The following display appears when you operate the READ-OUT softkey.





You can use the cursor keys to select the data type which is to be read out.



In the case of MAIN PROGRAMS, SUBROUTINES, TOOL OFFSETS and R PARAMETERS you can enter the area which is to be transferred. Enter the desired values in the appropriate input fields as described in Section 2.4.4 (Entry in input fields).

READ-OUT START

Operate the READ-OUT START softkey to initiate data output. The data being read out is displayed on the screen.

Note:

Use the STOP softkey to halt data output at any time and the READ-OUT START softkey to continue.

© Siemens AG 1990 All Rights Reserved

6.4 PLC program read-in/read-out

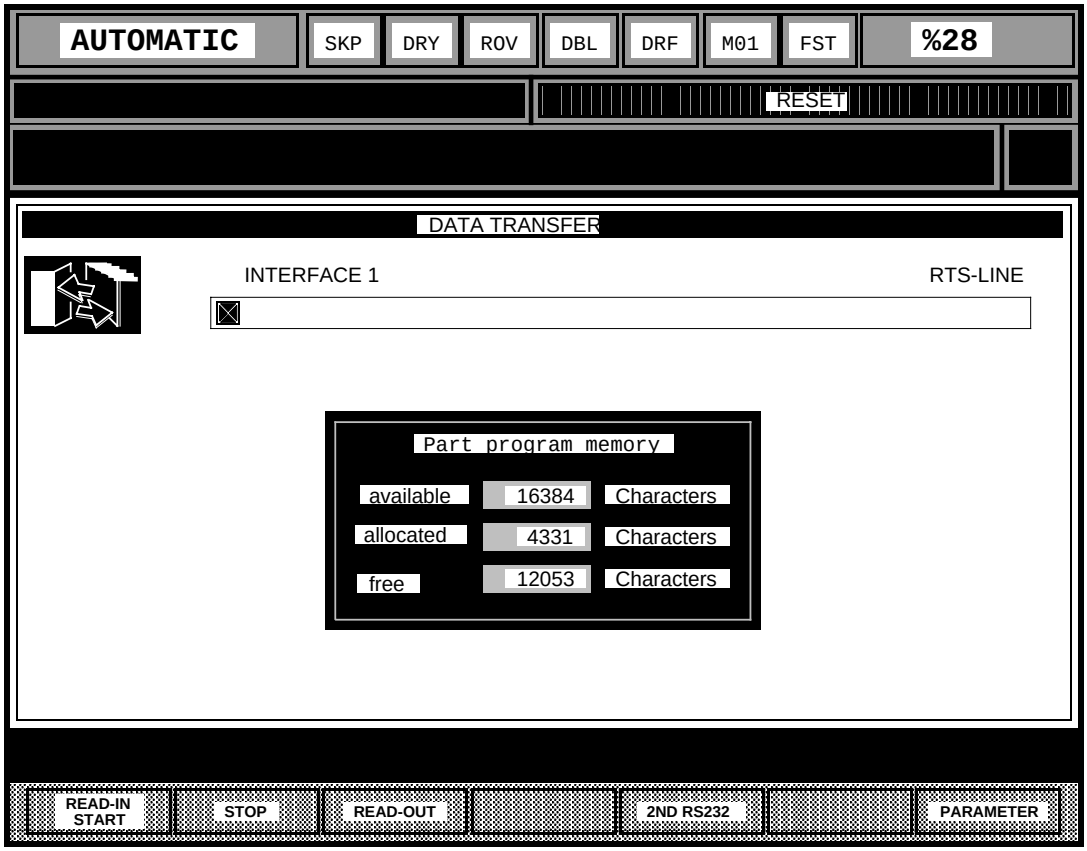
Sequence of operation

Prerequisite: The SINUMERIK 805 machine area is selected.

Use the OPERATING AREA softkey to select the SINUMERIK 805 data area.

DATA TRANSFER.

The following display appears when you operate the DATA TRANSFER softkey:

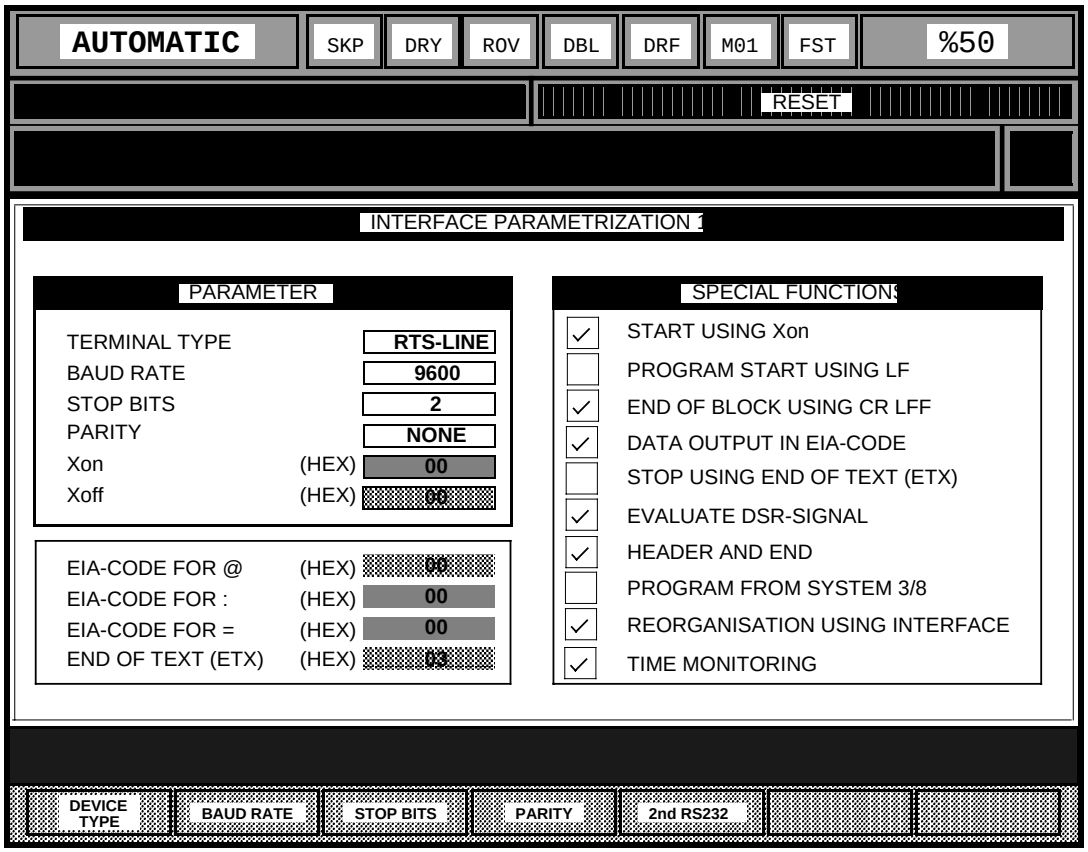


Interface 1 must be selected for PLC program read-in/read-out

© Siemens AG 1990 All Rights Reserved

PARAMETER

Operate the PARAMETER softkey to set or control the interface parameters. The following display appears:



© Siemens AG 1990 All Rights Reserved

The following settings can be selected using softkeys:

DEVICE TYPE	Device type:	PLC-PROG
BAUD RATE	Baud rate:	9600
STOP BITS	Stop bits:	2
PARITY	Parity:	none

After entering the interface parameters, operate the return key to return to the DATA TRANSFER basic menu.

READ-IN START

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

The interface is activated for PLC operation.

© Siemens AG 1990 All Rights Reserved

6.5 Interface parameters for various peripheral devices

For the purposes of data transfer between the SINUMERIK 805 and a peripheral device, the selected SINUMERIK 805 interface must be adapted to the 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 peripheral devices:

Parameters / Per device	Siemens programmer unit PG 750/685/675 20 mA Interface AG S5	Siemens programmer unit PG 750/685/675 RS 232 C Interface printer	Siemens programmer unit PG 615 U 20 mA	Siemens page printer PT 80 RS 232 C/20 mA	Siemens page printer PT 88 RS 232 C
TERMINAL TYPE	PLC-PROG	RTS-LINE	PLC-PROG.	RTS-LINE	RTS-LINE
BAUD RATE	9600	9600	9600	300	9600
DATA BITS	8	8	8	8	8
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					
Header and end		X		X	X
Program from system 3/8					
REORG using interface		X		X	X
Time monitoring		X		X	X

X = Parameter is active

© Siemens AG 1990 All Rights Reserved

Parameters / Per device	Siemens programming workstation PD.., PG RS 232 C	SINUMERIK System 800 NC-NC coupling 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
DATA BITS	8	8	8	8	8	8
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						
Header and end						
Program from system 318						
REORG using interface						
Time monitoring						

X = Parameter is active

© Siemens AG 1990 All Rights Reserved

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
DATA BITS	8	8	8
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			
Header and end			
Program from system 318			
REORG using interface			
Time monitoring			

X = Parameter is active

© Siemens AG 1990 All Rights Reserved

6.6 Setting data for defining interfaces

Interfaces 1 and 2 are defined using setting data.
An alteration of interface parameters for interface 1 or 2 in the PARAMETER menu corresponds to an alteration in the following setting data areas:

SD area:	Interface
SD no. 5010 to 5017	1
SD no. 5018 to 5025	2
SD no. 5026 to 5029	1 and 2

The setting data of both interfaces can be changed by reading in data using the data type SEA (see Section 5.2.12.1). The alteration only becomes active once the interfaces are started and is shown in the PARAMETER menu in plain text.

The tables below show the allocation of interface parameters to setting data bits as well as the setting data bits for the connection of peripheral devices.
Only those setting data numbers (SD no.) which are decisive for the connection of the device concerned to interface 1 or 2 are listed.

N**ote:**

The setting data areas listed above are not displayed in a table on the screen, but are contained in the data read-out (see Section 5.2.12.2).

© Siemens AG 1990 All Rights Reserved

SD no.		Setting data function							
Inter-face 1	Inter-face 2								
5010	**5018**	Device encoding input							
5011	**5019**	Transfer format input							
		Bit no. 7	Bit no. 6	Bit no. 5	Bit no. 4	Bit no. 3	Bit no. 2	Bit no. 1	Bit no. 0
		Stop bit	Stop bit	Parity bits	Parity bits	Baud rate	Baud rate	Baud rate	Baud rate
5012	**5020**	Device encoding output							
5013	**5021**	Transfer format output							
		Bit no. 7	Bit no. 6	Bit no. 5	Bit no. 4	Bit no. 3	Bit no. 2	Bit no. 1	Bit no. 0
		Stop bit	Stop bit	Parity bits	Parity bits	Baud rate	Baud rate	Baud rate	Baud rate
5014	**5022**	X_{ON} characters (DC start character)							
5015	**5023**	X_{OFF} characters (DC stop character)							
5016	**5024**	Special functions:							
		Bit no. 7	Bit no. 6	Bit no. 5	Bit no. 4	Bit no. 3	Bit no. 2	Bit no. 1	Bit no. 0
		Output without first Xon character	Progr. start using LF	Block end CR LF	Output in EIA code	Stop using end of text	Ready to start evaluate DSR signal	Output without header and end	Read in programs from SINU-MERIK systems 3,8
5017	**5025**	Special functions:							
		Bit no. 7	Bit no. 6	Bit no. 5	Bit no. 4	Bit no. 3	Bit no. 2	Bit no. 1	Bit no. 0
		Not assigned (no function)							Time monitor-ing switched off
5026	**5026**	EIA code for "@"							
5027	**5027**	EIA code for ":"							
5028	**5028**	EIA code for "end of text"							
5029	**5029**	EIA code for "="							

Description of interfaces

© Siemens AG 1990 All Rights Reserved

6.7 Assignment of the setting data for the connection of peripheral devices

Peripheral device	Ident. of the device type (CRT display)	SD no. Interface 1	SD no. Interface 2	Required setting data bits (binary code)
SINUMERIK Reader: T40, T50,T60 RS 232 C, 9600 baud	RTS-LINE	5010 5011	5018 5019	0000 0000 1100 0111
SINUMERIK WS 800 RS 232 C, 20 mA 9600 baud	RTS-LINE	5010 5011 5012 5013	5018 5019 5020 5021	0000 0000 1100 0111 0000 0000 1100 0111
Siemens Programmer unit PG 685/675/670 20 mA, 9600 baud Interface AG S5	PLC-PROG.	5010 5011	5018 5019	0000 0100 0000 0111
Siemens Programmer unit PG 675 RS 232 C, 1200 baud Interface printer	RTS-LINE	5010 5011 5012 5013 5016 5028	5018 5019 5020 5021 5024 5028	0000 0000 1100 0100 0000 0000 1100 0100 0010 1000 0000 0011
Siemens Programmer unit PG 615 U 20 mA, 9600 baud	PLC-PROG.	5010 5011	5018 5019	0000 0100 0000 0111
Siemens page printer PT80 20 mA RS 232 C, 300 baud	RTS-LINE	5010 5011 5012 5013	5018 5019 5020 5021	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 1990 All Rights Reserved

Peripheral device	Ident. of the device type (CRT display)	SD no. Interface 1	SD no. Interface 2	Required setting data bits (binary code)
Siemens Printer PT 88 RS 232 C, 9600 baud	RTS-LINE	5012 5013	5020 5021	0000 0000 1100 0111
SINUMERIK System 800, NC-NC Coupling, RS 232 C, 9600 baud	RTS-LINE	5010 5011 5012 5013	5018 5019 5020 5021	0000 0000 1100 0111 0000 0000 1100 0111
Siemens Programming Workstation PD..., PG... RS 232 C, 4800 baud	PD/PF	5010 5011 5012 5013 5014 5015	5018 5019 5020 5021 5022 5023	0000 0011 1100 0110 0000 0011 1100 0110 0001 0001 1001 0011
Sanyo cassette M2502U-ZE601 RS 232 C, 1200 baud	RTS-LINE	5010 5011 5012 5013	5018 5019 5020 5021	0000 0000 1100 0100 0000 0000 1100 0100
Teletype, ASR33 Full duplex 20 mA, 110 baud	RTS-LINE	5010 5011 5012 5013	5018 5019 5020 5021	0000 0000 1100 0000 0000 0000 1100 0000
Facit reader 4040, 4042 Reader/punch RS 232 C, 1200 baud	RTS-LINE	5010 5011 5012 5013	5018 5019 5020 5021	0000 0000 1100 0011
Facit reader 4030 RS 232 C, 1200 baud	RTS-LINE	5010 5011	5018 5019	0000 0000 1100 0100
Sommer terminal MDC-3 SNC cas- sette unit, RS 232 C, 9600 baud	RTS-LINE	5010 5011 5012 5013	5018 5019 5020 5021	0000 0000 1100 0111 0000 0000 1100 0111

© Siemens AG 1990 All Rights Reserved

6.8 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.
(RS 232(V.24))
6FC9 344-1Q. (20 mA)

Device data
Interface RS 232 (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
RS 232 (V. 24): no hardware setting
TTY: passive
Socket X9
Jumper 2 - 14 Receiver
3 - 13
6 - 10 Transmitter
7 - 9

© Siemens AG 1990 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 1 (20 mA current-loop 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

Siemens Programmer Unit PG 675

Interface **Printer**

Cable Order No.: 6FC9 344-1A.

Device data

Interface	RS 232 C
Transfer rate	1200 baud
Character format	1 start bit 8 data bits 2 stop bits

Operating conditions

Archiving of NC cycles 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 RS 232 C interface on the SINUMERIK.

© Siemens AG 1990 All Rights Reserved

Siemens page printer PT80

Cable Order No.: 6FC9 340-8C **.**
RS 232 (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 RS 232 C: 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
(RS 232 C)

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 1990 All Rights Reserved

7 Maintenance

7.1 Operating data

Modules/components carrying this warning symbol (yellow hand on a black triangle, with line through) are endangered by electrostatic charge.

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.

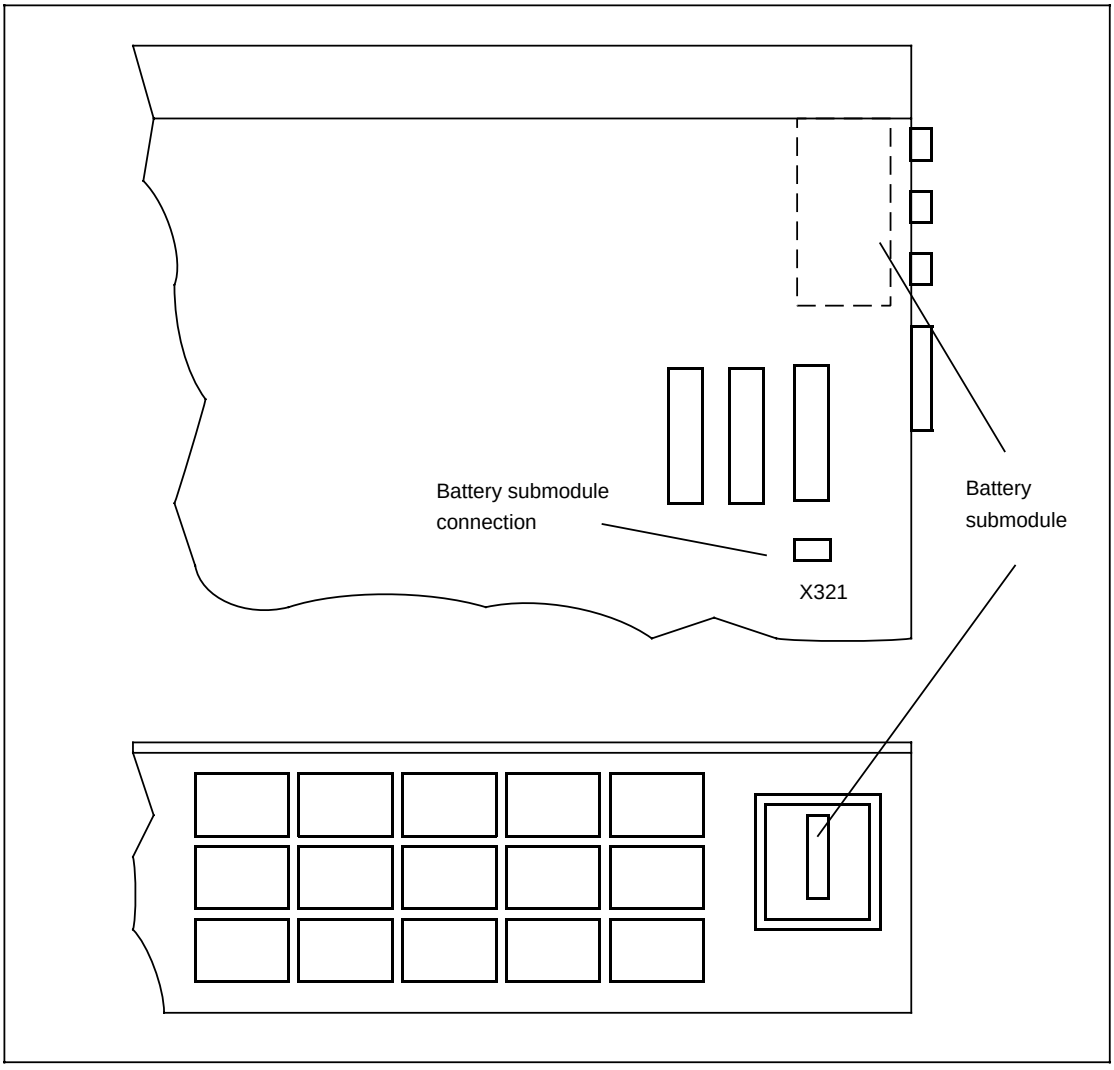
© Siemens AG 1990 All Rights Reserved

7.2 Battery submodule

If the voltage is switched off, the RAM is backed up by a battery voltage. The following data are maintained in the memory:

- Part programs
- Tool offsets
- Setting data
- Zero offsets
- Machine data (NC-MD and PLC-MD)
- R Parameter
- PLC user program
- User texts (PLC alarm texts and PLC message texts)

A battery submodule which is connected to the central processing unit via X321 backs up the RAM memory.



Battery submodule slot

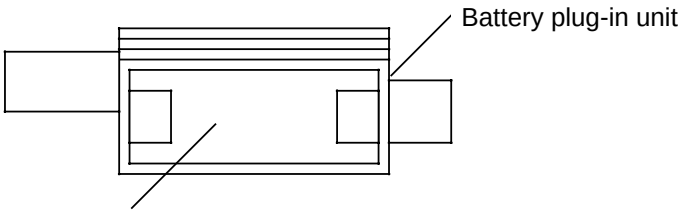
© Siemens AG 1990 All Rights Reserved

7.2.1 Replacing the battery

- The back-up time of the battery submodule is 1 year.
- Alarm 1 (battery alarm) is displayed when the battery should be replaced.
- Once alarm 1 has responded, the battery must be changed within 1 week!

Note:

The battery submodule uses 3 miniature cells (usual in trade) as shown below:



Batteries: 3x1,5 V miniature cells, LR6

1) The battery may only be replaced when the control is switched on!
2) When replacing the battery, ensure that the polarity is correct!
3) Please follow the regulations regarding the disposal of old batteries. If in doubt consult the appropriate authorities.

7.3 CRT display

The pictures on the CRT display can oscillate when the monitor is exposed to electromagnetic fields. Devices which generate electromagnetic fields, such as transformers, fans, electromagnetic switches, a.c. cables under power, etc. must be more than 300 mm away from the CRT display unit.

7.4 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
– greases and oils

© Siemens AG 1990 All Rights Reserved

8 Monitoring

8.1 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 and the drives 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 and PLC
- Voltage level of the backup battery
- System program memory
- User program memory

In addition, 64 PLC alarms and 64 PLC messages as well as 100 user cycle alarms are available to the PLC user. The texts for these are transferred via RS 232 C interface into the buffered memory area of the control.

8.2 Alarm numbers and groups/clearing alarms

The SINUMERIK 805 features the following alarm groups:

- NC alarms:
  - POWER ON alarms
  - RS 232 C alarms
  - RESET alarms (axis-specific)
  - RESET alarms (general)
  - Acknowledge alarms

- PLC alarms:
  - PLC system and programming alarms
  - PLC user alarms

- PLC messages:
  - PLC user messages

© Siemens AG 1990 All Rights Reserved

Type	Numbers	Groups		Acknowledged by:
NC alarms	1...15 40...99 132* 1)	Power On alarms		Switching the control off and on
	16 . . 39	RS 232 C alarms		1. Selecting data area 2. Operating DATA TRANSFER softkey 3. Operating STOP softkey
	100* . . 196*	axis-spec.	Reset alarms	Operating the RESET key on the operator keyboard
	2000 . . 2999	gen.		
	3000 . . 3081	Acknowledge alarms		Operating the QUIT softkey
	4000 . . 4999	Standard cycle alarms		
	5000 . . 5099	User cycle alarms		
PLC alarms	6000 . . 6063	User alarms		Operating the QUIT softkey
	6100 . . 6163	PLC system and programming alarms		Switching the control off and on
PLC messages	7000 . . 7063	User messages		No acknowledgement/clear necessary

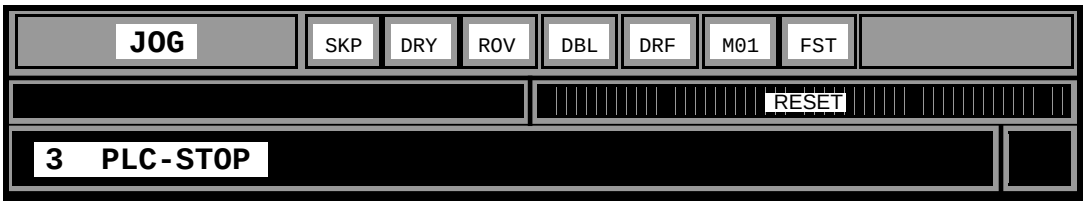
1): the asterisk * represents: "0" for alarms in axis 1
"1" for alarms in axis 2
"3" for alarms in axis 3
"3" for alarms in axis 4

© Siemens AG 1990 All Rights Reserved

8.3 Monitoring display on the screen

8.3.1 Message line for NC and PLC alarms and messages

Only the alarm with the lowest alarm number is displayed in the message line for NC or PLC alarms and messages.



Message line for NC or PLC alarms and messages

An alarm is displayed in the message line along with its alarm number and accompanying text (in example: alarm no. 3 (PLC-STOP).

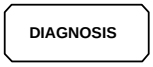
8.3.2 List of active alarms and signals

In the event of an alarm, several different alarms and/or messages can be recorded by the NC. A listing of all active alarms and messages can be selected as follows:

Sequence of operation

Prerequisite: The SINUMERIK 805 machine area has been selected.

Operate the OPERATING AREA key to select the data area.

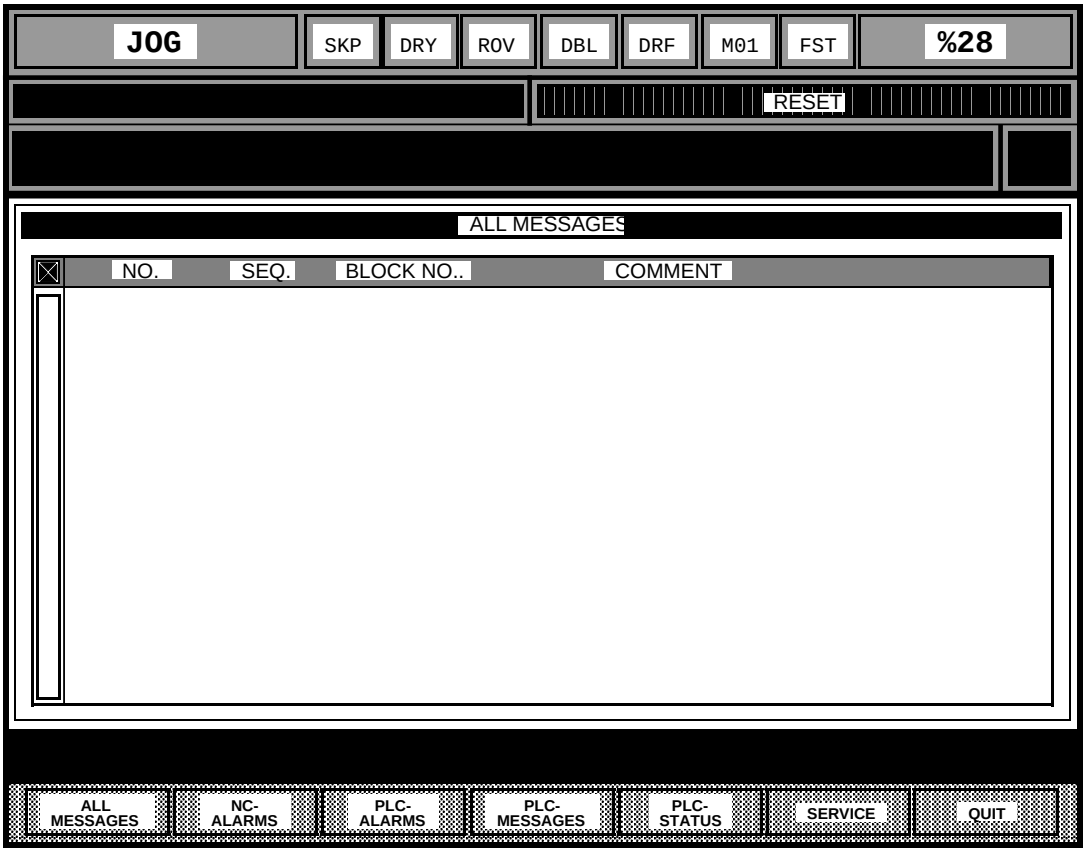


Operate the DIAGNOSIS softkey.

Note:

Explanations to display and DIAGNOSE softkey in Section 4.8.

© Siemens AG 1990 All Rights Reserved



Notes:

- When you operate the OPERATING AREA key, you return to the basic menu of the set operating mode in the machine area.
- The sequence no (SEQ.) shows the order in which the alarms have appeared. The sequence numbers start at 1 and finish at 99. The sequence number returns to 1 each time the control is turned on.
- In the case of alarms which occur whilst a program is being processed, the block number concerned is also displayed.
- The alarm number of the active alarm is displayed at the beginning of the line.

For a separate overview of the active NC alarms, PLC alarms or PLC messages, operate the following softkeys:

Softkey NC-ALARMS
Softkey PLC-ALARMS
or
Softkey PLC-MESSAGES in the MESSAGES basic menu

© Siemens AG 1990 All Rights Reserved

8.3.3 List of alarms

1	**Battery alarm! Do not switch off! Change battery!**
Cause:	Back-up battery voltage too low
Scan:	– On POWER ON – Cyclic
Explanation:	Replace battery (see Section 7.2.1). Battery voltage has dropped so low that backup of the user memory can only guaranteed for a short time.
Remedy:	Replace battery (see Instruction Manual), old batteries must be disposed of separately.
Caution!	Do ***not switch off*** the control, otherwise data will be lost.

2	**Operator panel error**
Cause:	Keyboard is not ready, cable damaged
Scan:	Cyclic

3	**PLC stop**
Cause:	PLC not operational
Scan:	Cyclic
Effect:	– Interlocking of NC START – Setpoint relay drop (setpoint 0) – Removal of NC READY 2. – Servo enable is removed in MD 156 after delay – has elapsed (servo enable relays drop out) – NC/PLC interface is put into the inactive state
Explanation:	Cyclic and alarm-controlled operation of PLC interrupted. It is not possible to traverse the machine.
Remedy:	Use the programmer to read out the cause of the interruption (USTACK).

4	**Invalid unit system**
Scan:	– On POWER ON – After modification of MD
Effect:	– Conversion factor taken as 1
Explanation:	In MD NC 5002 an illegal combination of measuring system unit (position resolution) and input unit system (conversion factor greater than 10) was selected.
Remedy:	Correct MD bit 5002 and switch control on/off.

© Siemens AG 1990 All Rights Reserved

7	**EPROM error**
Scan:	Cyclic
Effect:	– Interlocking of NC-START – Setpoint relay drop (setpoint 0) – Removal of NC READY 2 – Servo enable is removed in MD 156 after delay has elapsed (servo enable relays drop out)
Explanation:	– An error was detected whilst checking totals
Remedy:	– Notify service

8	**Incorrect axis/spindle assignment**
Scan:	After modification of MD
Effect:	– Interlocking of NC START – Setpoint relay drop (setpoint 0)
Explanation:	Removal of NC READY 2. An illegal assignment had been made in MD 200* or MD 400*. ***Correct:*** (0000) Axis/spindle not available at machine (only permissible with NC-MD bit 564*.7 = "0" (for axes) 0100 0200 0300 0400 0500
Remedy:	See Section 8 (MD description for MD 200* and MD 400*)

12	**Part program memory wrongly formatted**
Scan:	On POWER ON
Effect:	– Fault in RAM area – The part program memory size falls below minimum size permitted.
Remedy:	Format part program memory in initialization mode

13	**RAM error**
Scan:	On POWER ON
Effect:	Error in module RAM area
Remedy:	– Format user memory and format part program memory in initialization mode – Change CC

© Siemens AG 1990 All Rights Reserved

16	Parity error RS 232 C
Effect:	– RS 232 C transmission interrupted – Last block declared invalid
Explanation:	Alarm can only occur if parity is selected. The character commenced has incorrect parity. The alarm has nothing to do with character parity errors RS 232 C in ISO or EIA punched tapes (alarm 23)
Remedy:	– Check RS 232 C parameters – Check external device

17	Overflow error RS232 C
Effect:	– RS232 C transfer interrupted – Last block declared as invalid
Explanation:	The external device has sent a new character although the CNC has not yet processed the old one.
Remedy:	– Check RS 232 C interface parameters – Test external device

18	Frame error RS 232 C
Effect:	– RS 232 C transfer is interrupted – Last block declared invalid
Explanation:	– Number of stop bits is incorrect – Incorrect baud rate – Number of data bits is incorrect
Remedy:	– Check RS 232 C interface parameters – Test external device – Number of data bits: 7 data bits + 1 parity bit (set external device)

19	I/O device not ready RS232 C
Effect:	No files are read in
Explanation:	The DSR signal from the external device is LOW
Remedy:	– Start external device – Do not use DSR

© Siemens AG 1990 All Rights Reserved

22	**Time monitoring RS232 C**
Explanation:	– The CNC cannot output a signal for 60 sec. -- The external device blocks the CTS (clear to send) signal for longer than 60 sec -- The external device does not send DC1 within 60 sec when control signals (DC1 to DC4) are used. – The CNC has not received a signal for 60 sec.
Remedy:	– Check external device and switch on – Check cable and plug in. – Switch off time monitoring in parameter menu.

23	**Character parity error RS 232 C**
Cause:	Punched tape soiled or damaged
Effect:	– RS 232 C transmission interrupted – Last block declared invalid
Explanation:	Depending on definition of start of program "%" or "EOR", the NC automatically establishes the ISO or EIA code upon receipt of this character and thus character parity. When checking the following characters, it was established that one character does not have the set parity.
Remedy:	Check punched tape or data on diskette

24	**Invalid EIA character RS 232 C**
Effect:	– Data transfer interrupted – Last block declared invalid
Explanation:	EIA character was read in with the correct parity, however, the character is not defined in the EIA code.
Remedy:	Check punched tape: check EIA code for "@" and EIA code for ":"
Caution:	As the "-" character is not defined in the EIA code, the following data cannot be read in: – %TEA1 (NC machine data) – %RPA (R parameter) – %TOA (tool offsets) – %TEA2 (PLC machine data) – %ZOA (zero offset) – main programs and subroutines with R parameter calculations

© Siemens AG 1990 All Rights Reserved

26	**Part prog. block > 120 characters RS 232 C**
Cause:	The part program block that has been read in contains more than 120 characters. Only the actual stored characters are counted (no spaces, no CR, ...)
Effect:	– Data transfer interrupted – Last block is not stored
Remedy:	Divide block into 2 or more blocks

27	**Data input disabled RS 232 C**
Cause:	"Cycle lock" interface signal is active: NC, PLC machine data texts have been read in without password input.
Effect:	No data has been stored
Remedy:	– Reset Q 78.3 via PLC status – Enter password (scan appears when machine data is altered)

28	**Ring buffer overflow RS 232 C**
Effect:	– RS232 (V.24) transfer is interrupted – The last blocks are declared as invalid
Explanation:	Data transfer speed so high that more characters are transferred than can be processed by the NC. If the program is transferred again, the faulty program must be deleeted beforehand
Remedy:	– RTS signal has no effect on the input device (RTS causes the input device to STOP) – Transmission speed (baud rate) too high

29	**Block > 254 characters RS 232 C**
Cause:	The read-in block has more than 254 characters. All read-in characters (e.g. spaces) are included in the count.
Effect:	– RS 232 C transfer interrupted – Last block is not stored
Remedy:	Divide block into two or more blocks.

30	**Part program memory full RS 232 C**
Cause:	The maximum part program memory is full
Effect:	– RS 232 C transfer interrupted – Last block is not stored
Remedy:	Delete old programs and reorganize memory

© Siemens AG 1990 All Rights Reserved

31	**No further part program input possible (RS 232 C)**
Cause:	The maximum number of programs stipulated by machine data has been reached.
Remedy:	– Delete old programs and reorganize memory – Modify MD 8 and reformat part program memory Order: a. "INITIALIZATION" mode b. "Format user data" softkey c. "Format part program memory" softkey Old programs are also deleted!

32	**Data format error (RS 232 C)**
Cause:	– The permissible number of decades after an address is not correct – The decimal point occurs in the wrong place – Part programs and subroutines are not correctly defined or concluded (check heading!) – NC expecting an "=" character, which is however not correctly defined in the EIA code.
Effect:	Data transfer interrupted. Last block is not stored
Remedy:	Check the program to be read in

33	**Programs differ (RS 232 C)**
Cause:	Read-in and stored program not identical with program with same number
Effect:	No data is stored
Remedy:	Delete old program or rename old program
Explanation:	If a new program is read in which has the same program number as one already stored, both programs are compared. If the programs are different, an alarm is given.

34	**Operator error (RS 232 C)**
Cause:	Data transfer in NC has started and the PLC gives second signal
Effect:	No data is read in
Remedy:	Stop data input and restart

© Siemens AG 1990 All Rights Reserved

35	**SIEMENS reader error (RS 232C)**
Cause:	Error message output from SIEMENS tape reader
Scan:	Only when parameters for SIEMENS reader have been set
Effect:	– Data transfer interrupted – Last block not stored
Remedy:	– Restart data transfer – If error reoccurs: replace SIEMENS reader.

104*	**DAC limitation**
Scan:	Cyclic
Effect:	No direct effect. Absorbed by the following error alarm 156*
Explanation:	The DAC setpoint entered is higher than in MD 268* (max. DAC setpoint). It is not possible to increase the setpoint any further!
Remedy:	– Traverse more slowly – Check the actual values (encoder) – Check MD 268* – Check the drive actuator – Check MD 364* and MD 368*

108*	**Actual part position overrun**
Scan:	At each axis movement (also follow-up mode)
Effect:	– Interlocking of NC START – Setpoint relay drop (setpoint 0) – Removal of NC READY 2. – Servo enable is removed in MD 156 after delay has elapsed (servo enable relays drop out) – NC/PLC interface is put into the inactive state – Follow-up mode – Actual value of the machine is lost (incorrect position).
Explanation:	The actual part value is multiplied by the control. With rapid traverse of the axis the register overflows in the case of the fault. The reference point is lost.
Remedy:	– Reduce maximum speed – Check MD for variable incremental weighting (MD* 364 and MD 368*)

© Siemens AG 1990 All Rights Reserved

112*	**Standstill monitoring**
Cause:	– Incorrect position control direction – Mechanically clamped axis has been pushed out of position. – Fault on control device (actuator), at the tacho, motor, in the mechanics or NC measuring circuit hardware
Scan:	– On standstill – On clamping – On deceleration
Effect:	– Interlocking of NC START – Setpoint zero – Servo enable is removed in MD 156 after delay has elapsed (servo enable relays drop out) – Follow-up mode
Explanation:	The following error could not be cleared faster than the time entered MD 156. On clamping the limit defined in MD 212* was exceeded.
Remedy:	– MD 212* (clamping tolerance) must be greater than MD 204* (exact stop limit coarse) MD 156 (switch-off delay control enable) must be large enough to allow the following error of the axis entered to be cleared within the period entered.

116*	**Contour monitoring**
Scan:	When machining in AUTOMATIC, but ***not*** – when accelerating – when decelerating – at speeds lower than that in NC MD 336* (contour speed) – Interlocking of NC START
Effect:	– Setpoint 0 – The control enable is removed after the time stored in MD 156 -- has elapsed (control enable relay drops out) – Follow-up operation
Explanation:	The tolerance band NC MD 332* was exceeded at a speed greater than MD 336* When accelerating or decelerating the axis has not reached the new speed within the time defined by the K_v factor
Remedy:	– Increase the tolerance band in NC MD 332* – Check the KV factor – Check the optimization of the speed controller – Check drive actuator

© Siemens AG 1990 All Rights Reserved

132*	Control loop hardware
Scan:	Cyclic
Effect:	– Interlocking of NC START – removal of NC READY 2 – Setpoint relay drops out (setpoint 0) – The control enable is removed after the time delay MD 156 has elapsed (control enable relay drops out) – Follow-up operation
Explanation:	The measuring circuit differential signals – are not in phase – have a short-circuit to earth – are missing
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

136*	Measuring system dirty
Scan:	Cyclic
Effect:	Interlocking of NC START The processing of the current program is completed
Explanation:	In measuring systems with contamination signal (e.g. EXE), the measuring system signals a fault to the NC.
Remedy:	Check the measuring system

144*	Zero mark monitoring
	(available soon)

148*	Software limit switch plus
152*	**Software limit switch minus**
Scan:	At each axis movement
Effect:	– Interlocking of NC START – Setpoint 0
Explanation:	The alarm is only active after reference point approach Depending upon the PLC Interface Signal "2nd Software Limit Switch Active", the 1st or 2nd software limit switch has been approached
Remedy:	– Traverse away from the end limit switch in the opposite direction – Check MD 224*, 228*, 232*, 236*

© Siemens AG 1990 All Rights Reserved

156*	**Speed command value**
Scan:	Cyclic
Effect:	– Interlocking of NC START – Setpoint 0 – The control enable is removed after the delay in NC MD 156 has elapsed (control enable relay drops out) – Follow-up operation
Explanation:	– 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
Remedy:	– Check whether the value in NC MD 264* is higher than the value in NC MD 268* – Check the drive – Check the measuring system – NC neutral earthing? – Check drive actuator – Check the position control direction (command value/actual value reversed?)

160*	**Drift too high**
Scan:	Cyclic
Effect:	– Interlocking of NC START – "+" or  "-" sign is in the field – No traverse movement possible
Explanation:	The drift to be compensated by the NC itself has risen to over approx. 500 mV
Remedy:	– Execute drift compensation Operation: -- Select Service menu in data area -- Operate DRIFT COMPENSATION softkey for the axis concerned – Check that the drift has been correctly calibrated on the drive unit – Check drive actuator – Check earthing concept

168*	**Servo enable for traver. axis not received**
Scan:	At each axis movement
Effect:	– Interlocking of NC START – Setpoint 0 – The control enable is removed after the time stored in MD 156 has elapsed (control enable relay drops out) – Follow-up operation
Explanation:	The axis-specific control enable signal was removed by the PLC user program during traversing
Remedy:	Check the PLC program

© Siemens AG 1990 All Rights Reserved

172*	**Working area limit plus**
176*	**Working area limit minus**
Scan:	– When machining in AUTOMATIC – When traversing axes in JOG mode
Effect:	Interlocking in NC START – Setpoint 0
Explanation:	The working area limit in the setting data has been reached
Remedy:	– Check the working area limitation in the setting data – Check the program
Note:	– Working area limitation in JOG is only active when NC MD 5003 bit 6 is set

184*	**Stop behind reference point cam**
Scan:	On reference point approach
Effect:	– Interlocking of NC START – Setpoint 0 – Reference point not reached
Explanation:	During reference point approach the axis between the reference cams and measuring system zero mark was stopped
Remedy:	Restart reference point approach

2000	**Emergency stop**
Scan:	Cyclic
Effect:	– Interlocking of NC START – Setpoint 0 – Servo enable is removed in MD 156 after delay has elapsed (servo enable relays drop out) – Follow-up mode
Explanation:	The emergency stop signal is transmitted by the PLC to the NC
Remedy:	– PLC-STATUS check that Q 78.1 = 0 – Check whether EMERGENCY STOP cams have been approached or whether EMERGENCY STOP keys have been operated – Check PLC program:
Caution:	According to the regulations, the EMERGENCY STOP state must be selected not only from the control (software) but also from the hardware (using relay).

© Siemens AG 1990 All Rights Reserved

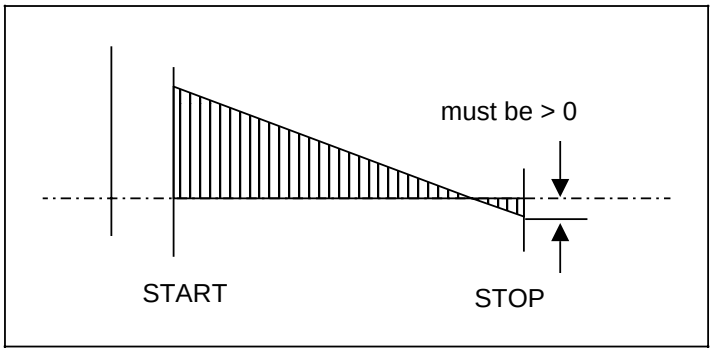
2031	Evaluation factor too high (MD 356*)
Scan: Effect: Remedy:	At each axis movement – Interlocking of NC START – Setpoint zero – Machining stop Check MD 388*

2032	Stop during threading
Effect: Explanation:	– Setpoint 0 – Interlocking of NC START During thread cutting, the feedrate per revolution was stopped and the thread destroyed.

2034	Speed reduction area
Explanation: Remedy:	The software pre-limit switch was overshot and the axis decelerated to the reduction speed – Check the program – MD 0 - pre-limit switch – MD 1 - speed beyond pre-limit switch (reduction speed)

2035	Programmed speed too high
Explanation: Remedy:	The programmed speed is higher than the contouring speed based on the maximum speed of the axes – Program lower contouring speed – Check MD 280* maximum contouring speed – Check MD 296* during reference point approach

© Siemens AG 1990 All Rights Reserved

2036	**G35 thread lead decr. error**
Scan: Explanation:	On thread cutting 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	**Progr. S value too high**
Explanation: Remedy:	The programmed spindle speed "S" is greater than "12 000" Program a lower spindle speed (S value is limited to "12000" in the control)

2039	**Reference point not reached**
Scan: Effect: Explanation: Remedy:	In AUTOMATIC/MDA after NC START Interlocking of NC START The reference point was not traversed to by all defined axes – Approach reference point with axes concerned – Set NC-MD 5004 bit 3 (NC START without reference point) ***Caution:*** Software limit switch not effective – Set NC-MD 560* bit 4 It is thus possible to disable the reference point approach axis-specifically for one or more specific axes. ***Caution:*** No software limit switches are active with these axes.

© Siemens AG 1990 All Rights Reserved

2040	Block not in memory
Effect: Explanation:	Interlocking of NC START – The block number sought in program with block search is not in the memory. – The programmed block number could not be found with the jump function in the given direction.

2041	Program not in memory
Effect: Explanation: Remedy:	Interlocking of NC START – The preselected program is not in the memory – The subroutine called in the main program is not in the memory Look at "Main programs" and "Subroutines" overview.

2042	Parity error in memory
Scan: Explanation: Effect: Remedy:	During machining in AUTOMATIC mode One or more characters have been deleted in the memory so that they can no longer be recognized (these characters are output as "?") Interlocking of NC START – Correct program in EDITOR or delete the complete block and re-enter it – If a large number of "?" appear, it could mean that the whole memory has been deleted in which case the battery should be checked and the part program memory reformatted

2046	Block > 120 characters
Scan: Effect: Explanation: Remedy:	During machining in AUTOMATIC Interlocking of NC START An "LF" has been corrupted in the memory, so that a block with more than 120 characters has arisen Insert "LF" without deleting the complete block

2047	Option not available
Effect: Explanation: Remedy:	Interlocking of NC START A function was programmed that is not included in the function set of the control Correct the program, check MD

© Siemens AG 1990 All Rights Reserved

2048	**Circle endpoint error**
Effect:	Interlocking of NC START
Explanation:	– The programmed circle endpoint does not lie on the circle – The endpoint is out by more than the value entered in MD 7 – No geometry in 1st block of the contour subroutine with L95 stock removal cycle
Remedy:	Correct program

2057	**Option not available**
Explanation:	– A thread was programmed in the memory using G33, G34 and G35, but this function has ***not*** been implemented in the control – Rev. feedrate G95 has been programmed
Remedy:	– Correct program – Check MD

2058	**Option not available**
Explanation:	– Three axes have been programmed simultaneously in the memory – A block has been programmed which causes the movement of three axes
Remedy:	– Correct program – Check MD

2059	**Program error G92**
Explanation:	– An invalid address character has been used – G92 programmed with address "P"
Remedy:	G92 is only permissible with address "S" (programmed spindle speed limit)

2060	**TO, ZO program error**
Meaning:	– A non-existent tool compensation number has been selected – The values in the selected zero offsets or tool compensations are too large – Tool offset type (P1) defined with 0

2061	**General program error**
Effect:	Interlocking of NC START
Remedy:	Select "Current block display" display and check block ***after*** current block.

© Siemens AG 1990 All Rights Reserved

2062	Feed missing/not programmed
Ursache:	– No F value programmed – F value too small (machine data) – Feed per rev. G95 programmed greater than 50 mm/min – No feed per rev. programmed
Remedy:	Programm feedrate correctly

2063	Thread lead too high
Effect:	Interlocking of NC START
Explanation:	A thread lead of more than 400 mm/rev (16"/rev) has been programmed
Remedy:	– Program lower thread leads – If necessary, run program on a machine with SINUMERIK 850 (max. speed 2000 mm/rev)

2064	Rounding err. for rotary axis wrongly prog.
Scan:	On traversing in AUTOMATIC/MDA mode
Explanation:	If you round to either a half or full degree on a rotary axis, the control will monitor whether the programmed positions correlate with the rounding.
Effect:	– Interlocking of NC START – Programmed path in block is ***not*** executed
Remedy:	– Program correct rotary axis position – Check MD 560* bits 2 and 3
Note:	In the JOG and 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

2065	Programmed position behind SW limit switch
Scan:	On traversing in AUTOMATIC/MDA mode
Effect:	– Interlocking of NC START – Programmed path ***not*** executed
Explanation:	The programmed block end point lies behind the software limit switch
Remedy:	– Correct program – Check MD 224*, 228*, 232*, 236* in dependence on PLC interface signal, "2nd software limit switch valid"

© Siemens AG 1990 All Rights Reserved

2066	**Thread lead increase/decrease too high**
Scan:	On traversing in AUTOMATIC/MDA mode
Effect:	Interlocking of NC START
Explanation:	A thread lead or pitch increase or decrease of more than 16 mm/rev (0.6"/rev) has been programmed
Remedy:	Program a smaller thread lead increase/decrease

2067	**Max. speed of one axis = 0**
Scan:	On traversing in AUTOMATIC/MDA
Effect:	Interlocking of NC START
Explanation:	An axis with a maximum speed of zero has been programmed in the block
Remedy:	Check MD

2068	**Prog. position behind working area limit**
Scan:	On traversing in AUTOMATIC/MDA mode
Effect:	– Interlocking of NC START – The programmed path has ***not*** been executed
Explanation:	The programmed block end point is in one or several axes behind the working area limit
Remedy:	– Check working area limit (plus and minus) – After working area limit in the program with G25/G26

2072	**Incorrect input value (Blue print prog.)**
Explanation:	Input for contour definition cannot be calculated

2073	**No intersection point (Blue print prog.)**
Explanation:	Calculation of the contour definition gives no intersection point with the values programmed

2074	**Incorrect angle value (Blue print prog.)**
Explanation:	– Angle 360° programmed – Angle value for the contour described not logical

© Siemens AG 1990 All Rights Reserved

2075	**Incorrect radius value (Blue print prog.)**
Explanation:	– Radius value too large – Radius value not allowed for the contour described

2076	**Incorrect G02/G03 (Blue print prog.)**
Explanation:	Direction of circle for the described contour ***not*** possible

2077	**Incorrect block sequence (Blue print prog.)**
Explanation:	Several block are needed for the calculation of the contour definition: – Block sequence not correct – Insufficient information (underdefined) **Example:** N10...B15 LF N20...G3 I20 LF

2078	**Incorr. input parameter (Blue print prog.)**
Explanation:	– Programmed parameter sequence not allowed – Parameter sequence not complete for described contour **Example:** N10...X60 B15 LF (Z axis missing) N20...X90 B10 LF

2081	**Program block with TRC/CRC not allowed**
Explanation: Remedy:	The following functions cannot be programmed with TRC/CRC (G41/G42) selected: G33, G34, G35, G58,G59, G92, M19 S..., – Pre-program G40 – Cancel with G41/G42 D00 (FRK/SRK)

2082	**CRC not determinable**
Explanation: Remedy:	Axes for the selected CRC plane do not exist – Check MD 548*, 550*, 552* (basic setting of G16) – Select correct plane with G16

© Siemens AG 1990 All Rights Reserved

2152	Spindle speed too high
Scan:	Only when MD 520* bit 2 is set (encoder available)
Explanation:	The spindle speed is higher than fixed in the machine data
Remedy:	– Program lower S value – MD 403* - 410* (max. spindle speed for the first to eighth gear stage) – MD 445* (tolerance band for max. spindle speed) – MD 451* (max. spindle speed) – Gear stage correctly selected by PLC? – G92 S... if v constant is incorrectly programmed

2153	Control loop error (spindle)
Scan:	Cyclic
Effect:	– Interlocking of NC START – Drop out of setpoint relay Setpoint 0 – Removal of NC READY 2 – Servo enable of spindles is removed in MD 4470 after delay has elapsed
Explanation:	As for alarm 132*
Remedy:	As for alarm 132*

2154	Measuring system dirty (spindle)
Scan:	Cyclic
Explanation:	In the case of measuring systems with a contamination signal, a fault is transmitted to the NC
Effect:	Interlocking of NC START
Remedy:	Check the measuring system

2155	Option not available
Scan:	On traversing in AUTOMATIC/MDA mode
Effect:	Interlocking of NC START
Explanation:	"M19 S..." is programmed in the part program although this function is not available
Remedy:	– Correct program – Retrofit M19

© Siemens AG 1990 All Rights Reserved

2171	**Approach not possible**
Explanation:	The control adds max. 1 axis per programmed plane. Approach is not possible if two axes are to be added.
Remedy:	– Check in the NC program that axis programming in approach block is complete – It is not possible to program a deselection block immediately after a selection block (tangent cannot be calculated).

2172	**Retract not possible**
Explanation:	See alarm 2171
Remedy:	– Check in the NC program that axis programming in approach block is complete – An approach movement must be programmed in G48 deselection movement (approach and withdrawal)

2173	**Wrong approach/retract plane**
Explanation:	The movements resulting from the selection/deselection of the function soft approach to and exit from contour refer to the selected plane
Remedy:	G16, G17, G18, G19 Check whether a plane change is programmed in the block after selection or in the deselection block

3000	**General program error**
Explanation:	In one block of the program, a general, not precisely definable programming error has been made **Example:** – An axis has been programmed which does not exist in the program – Incorrect interpolation number has been programmed
Remedy:	Check the faulty block in the part program The block number appears in the "Signals" display.

3001	**More than 5 geometry parameters programmed**
Explanation:	More than 5 geometry parameters, such as axes, interpolation parameters, radii, angles have been programmed in the block.
Remedy:	As for alarm 3000

© Siemens AG 1990 All Rights Reserved

3002	Polar radius programming error
Explanation:	In the block with polar/radius programming, no programming for: – angle – radius – coordinates for the centre point
Remedy:	As for alarm 3000

3003	Invalid address
Explanation:	– An address has been programmed which is not defined in the machine data – The axis names for initial setting plane G16 (MD 548*, 550*, 552*) do not correspond to the defined axis identifications (MD 568*)
Remedy:	– Like alarm 3000 – Correct machine data

3004	CL800 error
Explanation:	– @-function not implemented – Incorrect address after @ – Number of addresses after @ incorrect – Value in K, R or P not permitted – Number of decades too high – Decimal point not permitted – Jump destination incorrectly defined – System cell (NC-MD, PLC-MD, TO) not available – Bit number too large – Incorrect angle information for sine and cosine
Remedy:	– @ acc. to Programming Instructions – Only K, R and P are permitted as address – Jump destinations: forwards with "+" backwards with "-" – Check validity of given addresses – If necessary, select decode single block (DEC-SBL) and check program

3005	Blue print progr. error
Explanation:	The coordinates in the blue print programming have been defined so that no intersection point is given
Remedy:	As for alarm 3000

© Siemens AG 1990 All Rights Reserved

3006	Wrong block structure
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 other addresses as "X" or "F" – M19 has been programmed with other address as "S" – Incorrect or no interpolation parameters for G02/G03 (MD 304*)
Remedy:	As for alarm 3000

3007	Error in programming setting data
Explanation:	– G25/G26 has been programmed – G92 has been programmed with another address as "S" – M19 has been programmed with another address as "S"
Remedy:	As alarm 3000

3008	Subroutine error
Explanation:	– Subroutine call without number of passes "P" – M30 has been programmed as program end – M17 at program end is missing – The fourth nesting level has been called (Only 3 subroutine levels are possible with 805) – M17 has been programmed in main program
Remedy:	As for alarm 3000

3009	Program disabled
Explanation:	Not applicable to SINUMERIK 805

3010	Intersection error
Bedeutung:	This error can appear in connection with the stock removal cycle L95 when: – The contour program is programmed without G0, G1, G2, G3 – @ 714 is programmed 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 program of L95 stock removal cycle
Remedy:	As for alarm 3000

© Siemens AG 1990 All Rights Reserved

3011	Too many axes progr./axes programmed twice
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
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
Remedy:	As for alarm 3000

3016	External data input error
Effect:	Data transfer interrupted
Explanation:	For external data input from PLC to NC: – the code is incorrect – the value is too large – the dimension identifier is invalid – the option is not available
Remedy:	– Check PLC program – check NC-MD, PLC-MD

3018	Distance from contour too large
Scan:	After NC-START (Automatic)
Effect:	– Machining stops
Explanation:	The distance to circle contour is too large after repositioning (MD 9)
Remedy:	Observe MD 9, move closer to contour

3019	Option not available
Explanation:	The 2nd RS 232 C interface has been started either using the softkey or from the PLC even though no option is available for it.
Remedy:	– Data transfer via the 1st RS 232 C interface – Retrofit option C62 (2nd RS 232 C interface)

© Siemens AG 1990 All Rights Reserved

3020	**Option not available**
Explanation:	A function has been programmed which is not implemented in the control
Remedy:	– As for alarm 3000 – Retrofit option

3021	**TRC/CRC contour error**
Scan:	When TRC/CRC is selected ***NOT:*** – in selection block – in deselection block
Explanation:	The compensation calculation results in a traversing movement which is opposite to the one programmed

3072	**Alarm text not available**
	(Available soon)

3081	**CRC not selected for approach**
Explanation:	The function "soft approach to and exit from contour" is only possible when cutter radius compensation has been selected. G41/G42 D0 understood as selected.
Remedy:	Select CRC

4100	**No D number active**
Explanation:	CRC/TRC selected without a D number being specified within a standard cycle
Remedy:	Check part program

4101	**Tool radius = 0**
Explanation:	The cutter radius is given as 0 In the case of standard cycles, this leads to errors
Remedy:	Enter radius in D number

© Siemens AG 1990 All Rights Reserved

4102	Cutting radius too large
Explanation:	Using this cutter would lead to contour errors with some of the standard cycles
Remedy:	Program other cutter

4103	Tool too wide
Explanation:	The width of the grooving tool is too large for the standard grooving cycle

4120	Spindle direction not programmed
Explanation:	No spindle direction was programmed before the standard cycle was called up
Remedy:	Changing of part program

4121	Spindle not in tolerance range
Explanation:	In case the spindle speed fluctuation is too great when standard cycles are used
Remedy:	Check drive actuator, MDs

4140	Machined part diameter too small
Explanation:	The value given for the machined part diameter in the standard cycle parameter setting is too small
Remedy:	Check part program

4180	Option not available
Explanation:	The standard cycle which has been called up requires an option which is not available in the control
Remedy:	Retrofit option

4200	Check definition R (Nxxx)
Explanation:	A parameter setting has been incorrectly defined
Remedy:	Define parameter setting correctly

© Siemens AG 1990 All Rights Reserved

5000 ⋮ 5099	**User cycle alarm**
Explanation: Remedy:	The alarm has been called using @ 4c0 in the user cycle Check the cycle

6000 ⋮ 6063	**PLC user alarms**
Explanation: Remedy:	Initiation bit has been set in PLC user program Defined by manufacturer

6100	**Signal converter missing**
Cause: Effect: Remedy:	Load and transfer command for I/O device which is not available, e.g. L PB, T PB PLC-STOP Check I/O address and/or STEP 5 program

6101	**Illegal MC5 mode**
Cause: Effect: Remedy:	STEP 5 command cannot be interpreted PLC-STOP – Check and/or reload PLC program – Evaluate U-STACK

6102	**Illegal MC5 parameter**
Cause: Effect: Remedy:	Parameter type illegal (I, O, T, C, T) or parameter value illegal PLC-STOP – Check PLC program – Evaluate U-STACK

© Siemens AG 1990 All Rights Reserved

6103	Transfer to missing DB
Cause:	L DW and/or T DW without previously calling a data block (A DB)
Effect:	PLC-STOP
Remedy:	Check PLC program, U-STACK

6104	Substitution error
Cause:	Error whilst substituting a PLC command
Effect:	PLC-STOP
Remedy:	Check PLC program (U-STACK)

6105	Missing MC5 block
Cause:	Calling a block which is not in the control (OB, PB, SB, FB)
Effect:	PLC-STOP
Explanation:	e.g. OB 2 not available
Remedy:	Enter missing block.

6106	Missing data block
Cause:	Calling a data block which is not in the control
Effect:	PLC-STOP
Remedy:	Enter missing data block

6107	Invalid segment LIR/TIR
Cause:	LIR: segment nos. 0-A valid TIR: segment nos. 0-6 valid
Effect:	PLC-STOP
Explanation:	See LIR/TIR Programming Instructions
Remedy:	Correct program

6108	Invalid segment TNB/TNW
Cause:	Source: segment nos. 0 to A valid Destination: segment nos. 0 to 6 valid
Effect:	PLC-STOP
Explanation:	See TNB/TNW Programming Instructions
Remedy:	Correct program

© Siemens AG 1990 All Rights Reserved

6109	Overflow block stack
Cause:	Nesting depth > 12
Effect:	PLC-STOP
Explanation:	When a block calls itself
Remedy:	Correct program

6110	Overflow interrupt stack
Cause:	More than two U-STACK entries
Effect:	PLC-STOP
Explanation:	Cyclic program (OB 1) is interrupted by alarm program (OB 2) and alarm program interrupts itself
Remedy:	See processing delay OB 2, alarm 6162

6111	MC5 command STS
Cause:	STS command programmed in FB
Effect:	PLC-STOP
Explanation:	Immediate termination of STEP 5 program processing

6112	MC 5 command STP
Cause:	STP command programmed
Effect:	PLC-STOP
Explanation:	PLC-STOP after termination of STEP 5 program processing

6113	Illegal MC5 time/counter
Cause:	STEP 5 time or counter not available or not enabled by MD
Effect:	PLC-STOP
Remedy:	– Correct program – alter PLC-MD 6

6114	Function macro
Cause:	Error in an assembler function block (basic program)
Effect:	PLC-STOP
Remedy:	See Programming Instructions Function Macros

© Siemens AG 1990 All Rights Reserved

6115	System commands disabled
Cause: Effect: Remedy:	Programmed command LIR, TIR, TNB, TNW PLC-STOP Set PLC-MD 2003 bit 4

6116	MD 0000: Alarm byte no.
Cause: Effect: Remedy:	PLC-MD > 31 set PLC-STOP Correct MD

6117	MD 0001: CPU load
Cause: Effect: Remedy:	PLC-MD 1 > 20 % PLC-STOP Correct MD

6118	MD 0003: Alarm run time
Cause: Effect: Remedy:	PLC-MD 3 > 2500 µs PLC-STOP Correct MD

6119	MD 0005: Cycle time
Cause: Effect: Remedy:	PLC-MD 5 > 320 ms PLC-STOP Correct M

6121	MD 0006: Last MC5 time
Cause: Effect: Remedy:	PLC-MD 6 > 31 PLC-STOP Correct MD

© Siemens AG 1990 All Rights Reserved

6123	Invalid servo sampling interval
Cause:	NC-MD 155 > 10
Effect:	PLC-STOP
Remedy:	Correct MD

6124	Gap in MC5 memory
Cause:	No smooth follow on from valid to invalid blocks
Effect:	PLC-STOP
Remedy:	General reset and reload program

6125	Dual assigment of inputs
Cause:	An input address has been used twice (overlapping of two DMP submodule input areas)
Effect:	PLC-STOP dependent on PLC-MD 2003 bit 2
Remedy:	Check PLC-MD 10 - 19

6126	Dual assigment of outputs
Cause:	An output address has been used twice (overlapping of two DMP submodule output areas)
Effect:	PLC-STOP dependent on PLC-MD 2003 bit 2
Remedy:	Check PLC-MD 10 - 19

6127	Alarm byte missing
Cause:	Selected alarm input byte not available in hardware
Effect:	PLC-STOP
Remedy:	– Alter PLC-MD 0 – Set alarm routing for alarm byte

6130	Syncro error basic program
Cause:	Synchronization pattern for assembler function blocks no longer correct
Effect:	PLC-STOP
Remedy:	PLC GENERAL RESET, reload PLC program, if necessary

© Siemens AG 1990 All Rights Reserved

6131	Syncro error MC5 program
Cause: Effect: Remedy:	Synchronization pattern for STEP 5 program blocks no longer correct PLC-STOP PLC GENERAL RESET, reload PLC program

6132	Synchro error MC5 data
Cause: Effect: Remedy:	Synchronization pattern for STEP 5 data blocks no longer correct PLC-STOP PLC GENERAL RESET, reload PLC program

6133	Illegal block basic program
Effect: Remedy:	PLC-STOP Replace system software

6134	Illegal block MC5 program
Effect: Remedy:	PLC-STOP PLC GENERAL RESET, reload PLC program

6135	Illegal block MC5 data
Effect: Remedy:	PLC-STOP PLC GENERAL RESET, reload PLC program

6136	Summing error MC5 block
Effect: Remedy:	PLC-STOP PLC GENERAL RESET, reload PLC program

6137	Summing error basic program
Effect: Remedy:	PLC-STOP Replace system software

© Siemens AG 1990 All Rights Reserved

6138	No response from MPC
Cause: Effect: Remedy:	Cable break, incorrect submodule no. set or similar PLC-STOP dependent on PLC-MD 2003 bit 2 Check cable and rack

6139	MPC transfer error
Cause: Effect: Remedy:	Same submodule no. set for 2 DMP submodules PLC-STOP dependent on PLC-MD 2003 bit 2 Check submodule no.

6140	PLC-MD 10...19: DMP start address wrong
Cause: Effect: Remedy:	Incorrect input in PLC-MD 10 -19 PLC-STOP dependent on PLC- MD 2003 bit 2 Check PLC-MD 10 - 19

6149	Stop using softkey PG
Cause: Effect: Remedy:	Stop command using PG PLC-STOP – PLC-START using PG – power on.

6150	Timeout: User memory
Effect: Explanation: Remedy:	PLC-STOP (S5 progr.) Evaluate detailed error coding, see Programming Instructions

6152	Timeout: LIR/TIR
Cause: Effect: Explanation: Remedy:	Access to addresses which are not available PLC-STOP See Programming Instructions Check segment and offset address. Hardware available?

© Siemens AG 1990 All Rights Reserved

6153	Timeout: TNB/TNW
Cause: Effect: Explanation: Remedy:	Incorrect programming or use of TNB/TNW PLC-STOP – See Programming Instructions – Check source and destination address for validity – Do addresses exist?

6154	Timeout: L PB/L PW/T PB/ T PW
Cause: Effect: Remedy:	Load and transfer command to I/O which has failed PLC-STOP Check I/Os and/or replace modules

6155	Timeout: Substitution command
Effect: Remedy:	PLC-STOP Check PLC program

6156	Timeout: Not interpretable
Cause: Effect: Remedy:	Acknowledgement timeout of system program not definable PLC-STOP – Evaluate detailed error diagnostics – PLC GENERAL RESET, reload PLC program

6157	Timeout: IN FB/JC FB
Cause: Effect: Remedy:	In the memory resident function macros, addresses are being accessed which are not available PLC-STOP Check hardware!

6158	Timeout on I/O transfer
Cause: Effect: Explanation: Remedy:	Central I/O no longer signalling PLC-STOP Central I/O recorded on start-up. No alarm when the + 24 V supply fails! Replace central controller

© Siemens AG 1990 All Rights Reserved

6159	Runtime exceeded step 5 program
Cause:	The max. runtime in PLC-MD 1 has been exceeded
Effect:	PLC-STOP, dependent on PLC-MD 2003 bit 1
Explanation:	Evaluate DB diagnostics!
Remedy:	– Increase MD 1 CPU load increases – Set MD 2003 bit 6 Cycle time increases – Time optimization of PLC program

6160	Runtime exceeded 0B 2
Cause:	The max. runtime in PLC-MD has been exceeded
Effect:	PLC-STOP, dependent on PLC-MD 2003 bit 0
Explanation:	Evaluate DB diagnostics!
Remedy:	– Increase MD 3 – Time optimization of OB 2

6161	Cycle time exceeded
Cause:	The max. runtime in PLC-MD 5 has been exceeded
Effect:	PLC-STOP
Explanation:	**Note:** bit commands only have a processing time of approx. 1 µs!
Remedy:	– Increase MD 5 – Time optimization of PLC program

6162	Processing delay OB 2
Cause:	The alarm program (OB 2) has interrupted itself
Effect:	PLC-STOP, dependent on PLC-MD 2003 bit 0
Explanation:	Evaluate DB diagnostics
Remedy:	Time optimization of OB 2, i.e. reduce the processing time of the alarm program!

6164	DMP protec. function output (short-circuit)
Cause:	Short-circuit at a DMP submodule output
Remedy:	Evaluate DB 1 diagnostics

© Siemens AG 1990 All Rights Reserved

6165	DMP 24V supply for logic not OK
Cause: Remedy:	Failure of 24V supply or shortfall of other operating point (15V) Check power supply

6166	DMP overtemperature (> 63 °C)
Cause: Remedy:	Overtemperature in DMP submodule Evaluate DB 1 diagnostics

7000 ⋮ 7063	PLC user operating message
Cause:	Initiation bit has been set in PLC user program

© Siemens AG 1990 All Rights Reserved

9 Appendix

9.1 Softkey function dictionary

Softkey	Significance	Section
2.V24	Interface selection (Interface 1 and 2 (option))	4.7
ACCEPT DATA	Alternate between insert and overwrite mode	4.3.1
ACTIVE BLOCK	Display of the active block contents	5.3.2
ALL MESSAGES	Display of all active NC resp. PLC alarms and PLC messages	4.8
BAUD RATE	Set baud rate	6.2
BLOCK SEARCH	Block search managesthe operation start at a user selectable part of the the program. During block search the same calculations like in normal operation run are made but there is no axis movement.	5.3.4
CLEAR	The program seleted by cursor or by entering the program number can be deleted.	4.3.8
CLEAR AREA	Clear an area which can be input	4.3.1
COPY	Copy the selected program onto a new program number	4.3.7
DATA PARAMETER	Display of R parameters	5.2.5
DATA TRANSFER	Selection of data transfer menu	4.7
DELETE BLOCK	Delete a block selected by cursor	4.3.1
DEVICE TYPE	Set interface type	6.2
DIAGNOSIS	Selection of NC recorded alarms and messages	4.8
DRIFT COMP.	Drift compensation for the selected axis	4.8.2
INITIALISATION	Selection of initialisation mode for installation	see Installation Guide
INSERT BLOCK	Insert a new block after the block selected by cursor	4.3.1

© Siemens AG 1990 All Rights Reserved

Softkey	Significance	Section
JOG	Selection of JOG mode	3.2.1
JOG-INC	Selection of JOG-INC submode	3.3
JOG-REF	Selection of JOG-REF submode	3.5
JOG-REPOS	Selection of JOG-REPOS submode	3.4
MAIN PROGRAM	Display of ”MAIN PROGRAM DIRECTORY”	4.3.5
MEMORY ALLOCATION	Display of the user program memory (part program memory), the free and allocated memory	4.3.4
MODIFY PROGRAM	Edit the program selected by cursor	4.3.1
NC ALARMS	Display of the active NC alarms	4.8
NC-MD BITS	Selection of the desired data type	4.4.4
NC-MD VALUES	Selection of the desired data type	4.4.3
NEW PROGRAM	Create a new part program	4.3.2
OPERATOR PROMPTING	Parameter support for subroutine programs	4.3.1.1 5.3.11.1
OVERRIDE BLOCK	Display of part program block with possibility to edit	5.3.14
OVERSTORE	Entry of auxiliary or additional functions, spindle speed and tool number into temporary storage	5.3.12
PARAMETER	Selection of the menu for interface parameter assignment	6.2
PARITY	Set parity selection for data transfer	6.2
PLC ALARMS	Display of active PLC alarms	4.8
PLC MESSAGES	Display of active PLC user messages **Note:** PLC user alarms resp. messages must have been stored by the manufacturer as a sub-routine program.	4.8

© Siemens AG 1990 All Rights Reserved

Softkey	Significance	Section
PLC STATUS	Information about PLC current state of inputs, output, flags etc.	4.8.1
PLC-MD	Selection of the desired data type	4.4.5
PRESET	Set actual value	5.3.13
PROGRAM	Branch into area ”part programs”	5.2.8.1
PROGRAM CONTROL	Modify the sequence of a part program	5.3.5
R PARAM	Selection of the desired data type	4.4.1
READ-IN START	Start of data input via selected interface	4.7
READ-OUT	Selection of ”data output” display	4.7
RELOCATE	The selected program is pushed to the end of the memory for editing	4.3.10
RENAME	The selected program is given another number	4.3.9
REORGANISE	Reorganisation of the part program memory	4.3.11
SEARCH	Key in a particular data parameter	4.4.1
SEARCH	Key in the program number of a particular part program	4.3.6
SELECT PROGRAM	Selection of part program for execution	5.3.1
SERVICE		4.8.2
SETTING DATA	Selection of the desired data type	4.4.2
START	Activation of the selected softkey function	5.3.4
STOP	Interrupts the data transfer during read-in/read-out	4.7
STOP BITS	Set the stop bit number	6.2
SUBROUTINE PROGRAMS	Alternate to ”SUBROUTINE DIRECTORY”	4.3.5

© Siemens AG 1990 All Rights Reserved

Softkey	Significance	Section
TOOL OFFSET	Display of the tool offset window	5.2.4
UNDO MOD.	Reverse the recent modifications in the block. This is only possible if the INPUT key has not yet been operated.	4.3.1
VERSION	Display of system screen	1.2
ZERO OFFSET	Display of an input window for zero offset	5.2.6.1

© Siemens AG 1990 All Rights Reserved

9.2 Index

9.2.1 Abbreviations

Abbreviations	Significance
ASM	User memory module
BCD	Decimal numbers in binary code (**B**inary **C**oded **D**ecimales)
CLF	Delete statement (**Cl**ear **F**ile), during read-in
CPU	**C**entral **P**rocessing **U**nit
DAU	Digital-analog converter
DBL	Decoding single block
DIO	Data transfer display (**D**ata **I**nput/**O**utput)
DRF	DRF shift
DRY	Test run feedrate
EIA-Code	Special tape code, hole number per character always odd
EOB	Block end mark for EIA code data
EOR	Mark for Program number (for EIA code)
EPROM	Program memory with resident program
EZS	Input temporary storage
FRK	Cutter compensation
H	Auxiliary function
INC	Operation mode ”Increment” (**Inc**rement)
ISO-Code	Special tape code, hole number per character always even
K1	Channel 1, main channel
K2	Channel 1, auxiliary channel
L	Subprogram no.
LED	Light emiting diode display (**L**ight **E**mitting **D**iode)
LF	Block end mark for ISO code data
M	Additional function
M01	Programmed stop
MD	Machine data

© Siemens AG 1990 All Rights Reserved

Abbreviations	Significance
MDA	Manual data automatic
MPF	Part program (**M**ain**P**rogram **F**ile)
N	Block number
NC	**N**umerical **C**ontrol
NV	Zero offset
P	Pass count
PCA	PLC alarms
PCP	PL program in machine code (**P**rogrammable **C**ontrol **P**rogram)
PLC	Interface control (**P**rogrammable **L**ogic **C**ontrol)
Pr. Halt	Display of a programmed run interruption
RAM	Program memory with read and write accesses
ROV	Rapid traverse override
RPA	R parameter with value assignment (**R P**arameter **A**ctive)
S	Spindle speed
SD	Setting data
SEA	Addresses with value assignments (**S**etting Data **A**ctive)
SKP	Skip block
SPF	Subroutine (**S**ub **P**rogram **F**ile)
SRK	Cutter radius compensation
SW-Endschalter	Software position switch
T	Tool number
TEA1	NC machine data (**T**esting Data **A**ctive)
TEA2	PLC machine data (**T**esting Data Active)
TOA	Tool offsets (**T**ool **O**ffset **A**ctive)
WZK	Tool offset
ZOA	Zero offset (**Z**ero **O**ffset **A**ctive)

© Siemens AG 1990 All Rights Reserved

9.2.2 Useful definitions

Definitions	Section
Absolute actual value	4.8.2
Absolute actual value	4.8.2
Actual position	2.1.1
Actual value system	4.4.2
Block search	2.1.1
Block search	5.3.4
Cleaning	7.4
Command value	4.8.2
Contour deviation	4.8.2
Cutting speed	4.8.2
Data area	2.4.3
Data input/output	4
Delete block	4.3.8
Delete input	4.3.8
Device setting data	6.7
Device-connection data	6.8
Differential resolver function (DRF)	5.3.7
Edit key	2.1.1
Entry field	2.1.1
Feedrate offset	2.2.1
Feedrate start/stop	2.2.1
Following error	4.8.2
Handwheel	2.2.1/3.3.2
Insert mode	4.3.1
Installation	see Installation Guide
Interfaces	6
Jog operation	5.3.10
Key in block/key in character	4.3.1
Machine area	2.4
Maintenance	7

© Siemens AG 1990 All Rights Reserved

Definitions	Section
Manual input/Automatic	3.6
MDA	3.6
Menu tree	2.4
Mode menu trees	2.4.2
Modify block	4.3.2
Monitoring	8.3.3
Operating mode overview	3.1.1
Operating mode selection	3.1.2
Operating modes	3
Operating sequences, operation	5.3
Operating sequences, preparation	5.2
Part actual value	4.8.2
Part command value	4.8.2
Position actual value	4.8.2
Program control	5.3.5
Program editing	4.3.1
Program interruption	5.3.3
Program management	4.3.11
Program memory	4.3.4
Program stop**/**Program start	5.3.3**/**5.3.1
R parameter	4.4.1
Reference point start	5.2.2
Replace mode	4.3.1
Repositioning	5.3.10
RESET	3.1.3
Scratch	5.3.17
Search	4.3.6**/**4.4.1
Self retentive	5.3.10
Set actual value	5.3.13
Setting data	4.4.2
Single block	2.1.1

© Siemens AG 1990 All Rights Reserved

Definitions	Section
Softkey functions	9.1
Softkeys	2.3
Speed actual value	4.8.2
Speed command value	4.8.2
Spindle speed	2.2.1
Spindle stop**/**Spindle start	2.2.1
Switch off	1.2
Switch on	1.2
Tool offset	4.5
Tool parameter	5.2.4
Tools	5.2.4.1
WZ parameter	5.2.4
Zero offset	4.6

© Siemens AG 1990 All Rights Reserved

Siemens AG

AUT V230
P.O. Box 48 48
W-8500 Nuremberg 1
Federal Republic of Germany

Suggestions

Corrections

For Publication/Manual:

SINUMERIK 805
Software Version 3

Operator's Guide

Order No.: 6ZB5 410-0EE02-0BA1
Edition: January 1991

From:

Name

Company/Dept.

Address

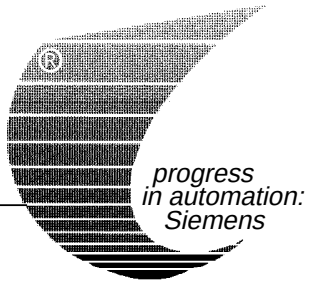
Telephone /

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 48 48, W-8500 Nuremberg 1
Federal Republic of Germany

© Siemens AG 1990 All Rights Reserved
Subject to change without prior notice



Siemens Aktiengesellschaft

Order No.: 6ZB5 410-0EE02-0BA1
Printed in the Federal Republic of Germany
251/211248 BA 05910.3