SIEMENS

SINUMERIK 810G
Basic Version 3
Software Version 1

Operator's Guide 05.92 Edition

User Documentation

SINUMERIK 810G
Basic Version 3
Software Version 1

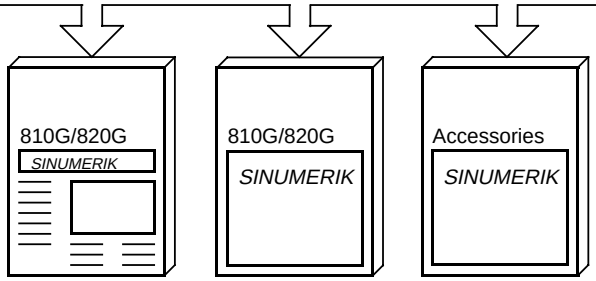
Operator's Guide

User Documentation

May 1992 Edition

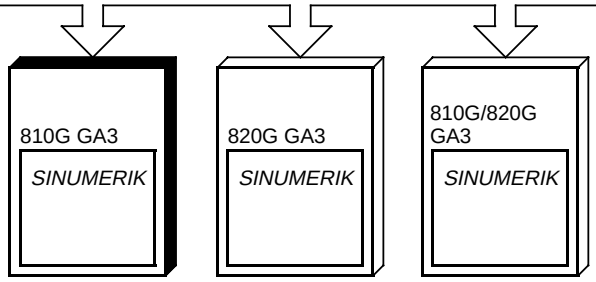
SINUMERIK 810G/820G GA3

General Documentation



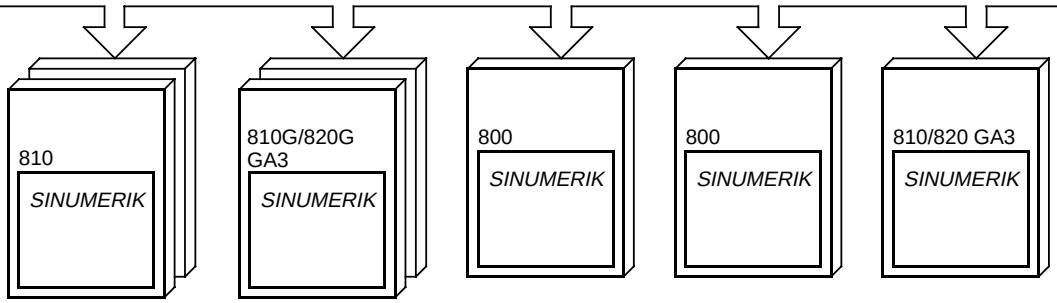
Technical Data | Catalog NC 25 | Catalog NC 90

User Documentation



User's Guide | User's Guide | Programing Guide

Manufacturer Documentation



Instruction Manual
- SIN 810
- SIN 820

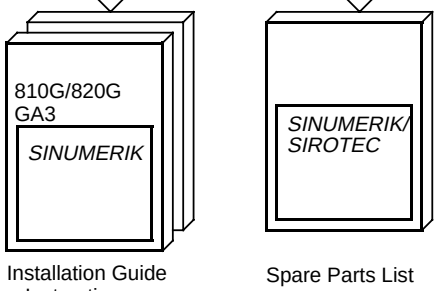
Interface:
- Signals
- Cables and Connections (T/M)

Universal Interface

SINUMERIK WS 800A CL800 Language

PLC Programming

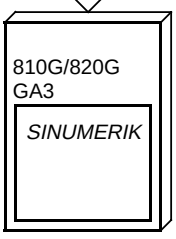
Service Documentation



Installation Guide
- Instructions
- Lists

Spare Parts List

User, Manufacturer and Service Documentation



Grinding Cycles, Vers. 2

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
05.92	6ZB5 410-0HQ02-0BA0	**A**

Preliminary Remarks

Notes for the reader

The SINUMERIK documentation is organized in three parts:
- User documentation
- Manufacturer documentation and
- Service documentation

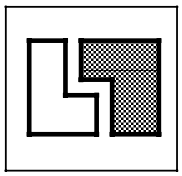
This Operator's Guide has been written for machine tool **users**.

The Operator's Guide explains:
- The configuration of the control
- The operating elements:
  - CRT display with softkeys
  - Keyboard and display panel of the control
  - Keys and switches on the external machine control panel
- Operating sequences
- Data interfaces and interfacing to the machine
- Diagnostics and maintenance

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

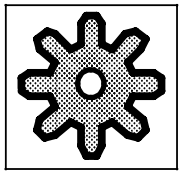
More information on other SINUMERIK 810G publications (or SINUMERIK 810 in general) and on publications which are available for all SINUMERIK controls (”Universal Interface”, ”Measuring Cycles”, . . .) can be obtained from your local **Siemens branch office**.

Technical comments



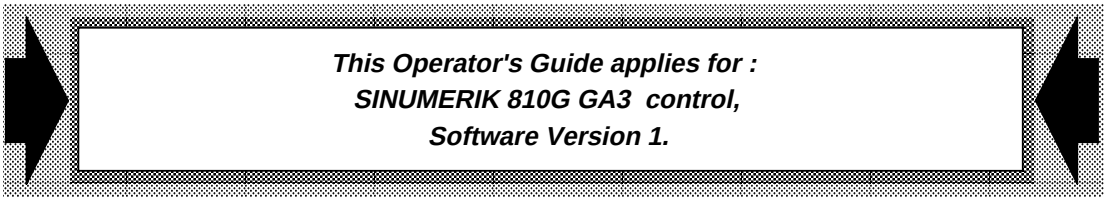
Occasionally in this documentation you will come across the symbol shown on the left and a reference to an Order Code.

This is intended to indicate that the function described is only capable of operating if the controller includes the option shown.
An overview of the possible Order Codes for the SINUMERIK 810G GA3 is provided in this Operator's Guide, Section 7.2.



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

It is possible that other functions which are not described in this documentation can 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.



Contents

1 Design

1.1 Product

The SINUMERIK 810G GA3 is a microprocessor-controlled CNC continuous-path control system for compact machine tools.

SINUMERIK 810G GA3 without machine control panel

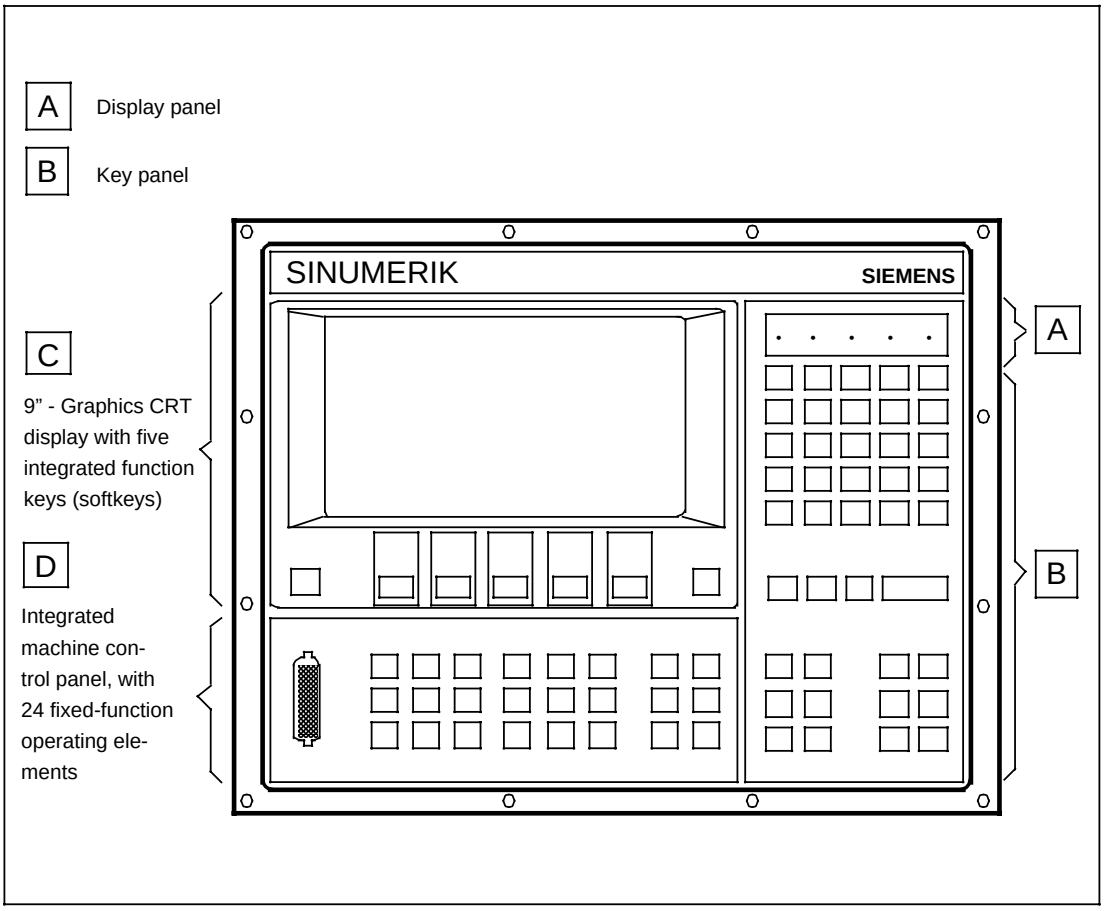
- The SINUMERIK 810G GA3
  is used primarily to control grinding and special-purpose machines
- Programming can be either computer-aided or manual
- Operation:
  – Softkeys for selecting different softkey functions
  – 9” graphics screen
  – Address /numerical keyboard and function keys
- Screen displays provide information in plain text, e. g. covering:
  – current NC operating modes
  – setpoint/actual values
  – NC and PLC alarms ...
- Graphics displays aid the programmer when entering programs at the machine.
- ”Blueprint programming” is available for quicker programming of complex contour elements.
- Entered programs can be simulated graphically.
- The SINUMERIK 810G GA3
  can process 9999 main programs and 9999 subroutines
  200 main programs and subroutines can be stored simultaneously in the memory.

1.2 Configuration

1.2.1 SINUMERIK 810G GA3 with integrated machine control panel

The SINUMERIK 810G GA3 with an **integrated** machine control panel incorporates in a single unit:

- Display panel
- Key panel
- 9” Graphics CRT display with integrated function keys (softkeys)
- Integrated machine control panel with 24 fixed-function operating elements.

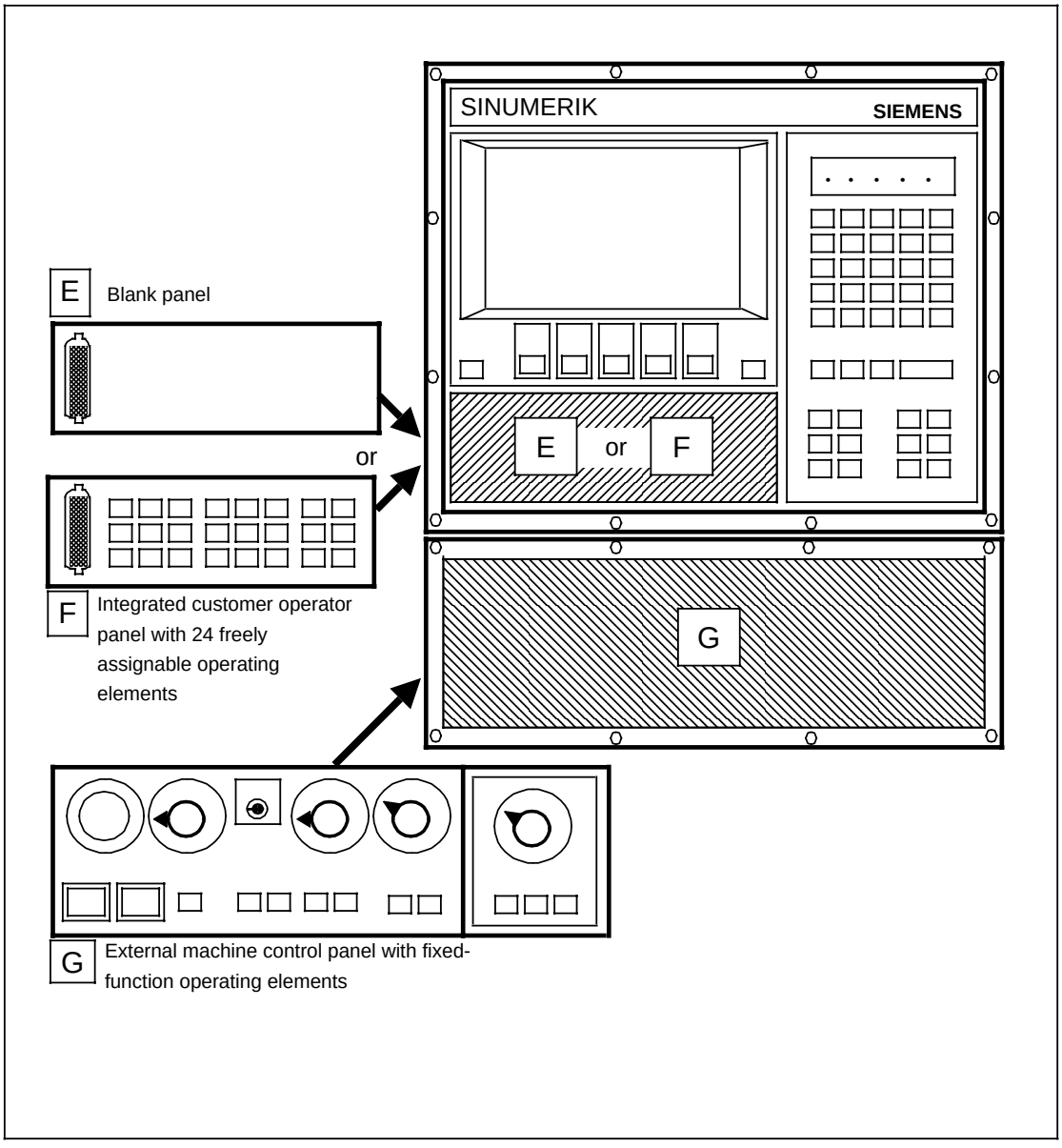


SINUMERIK 810G GA3 with integrated machine control panel

1.2.2 SINUMERIK 810G GA3 with external machine control panel

The SINUMERIK 810G GA3 with an **external** machine control panel incorporates in a single unit:

- Display panel, key panel, 9” graphics CRT display as described in 1.2.1
- Blank panel E
- or integrated customer operator panel with 24 freely assignable operating elements F
- External machine control panel with fixed-function operating elements G

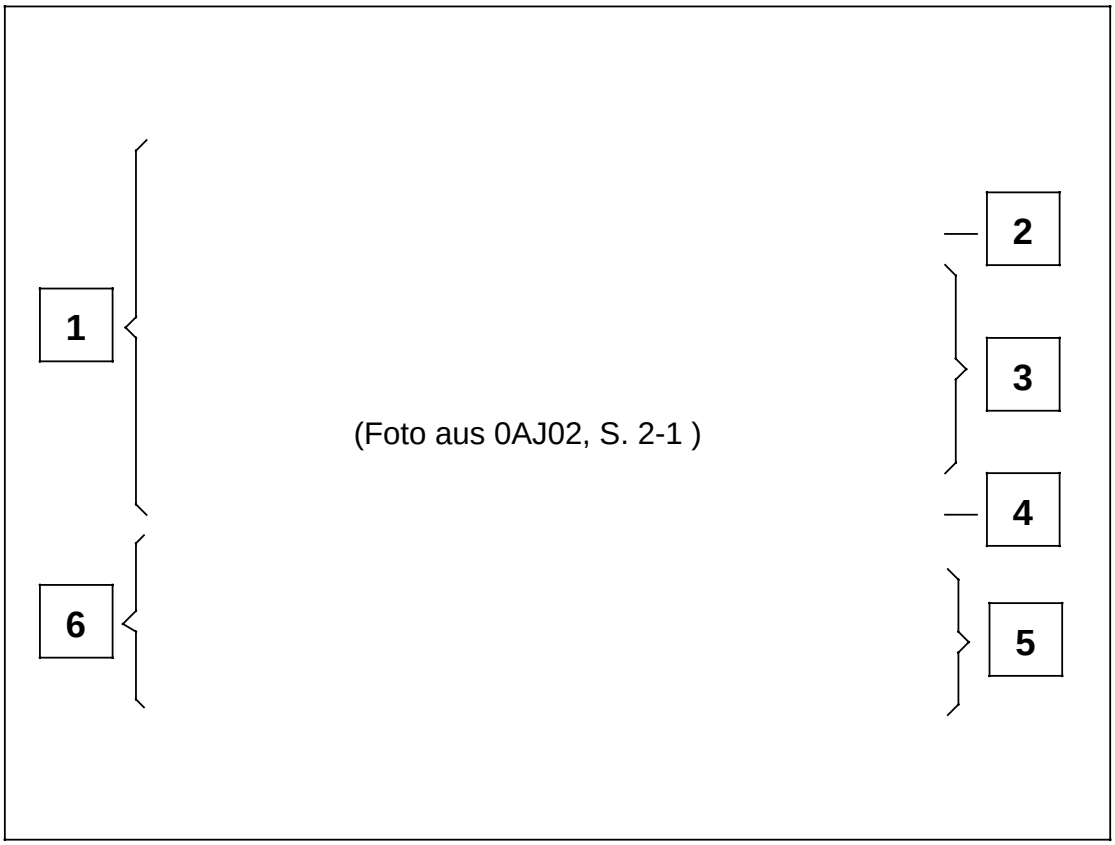


SINUMERIK 810G GA3 with external machine control panel

2 Operation

2.1 Operating elements

2.1.1 SINUMERIK 810G GA3 operator interface with integrated machine control panel

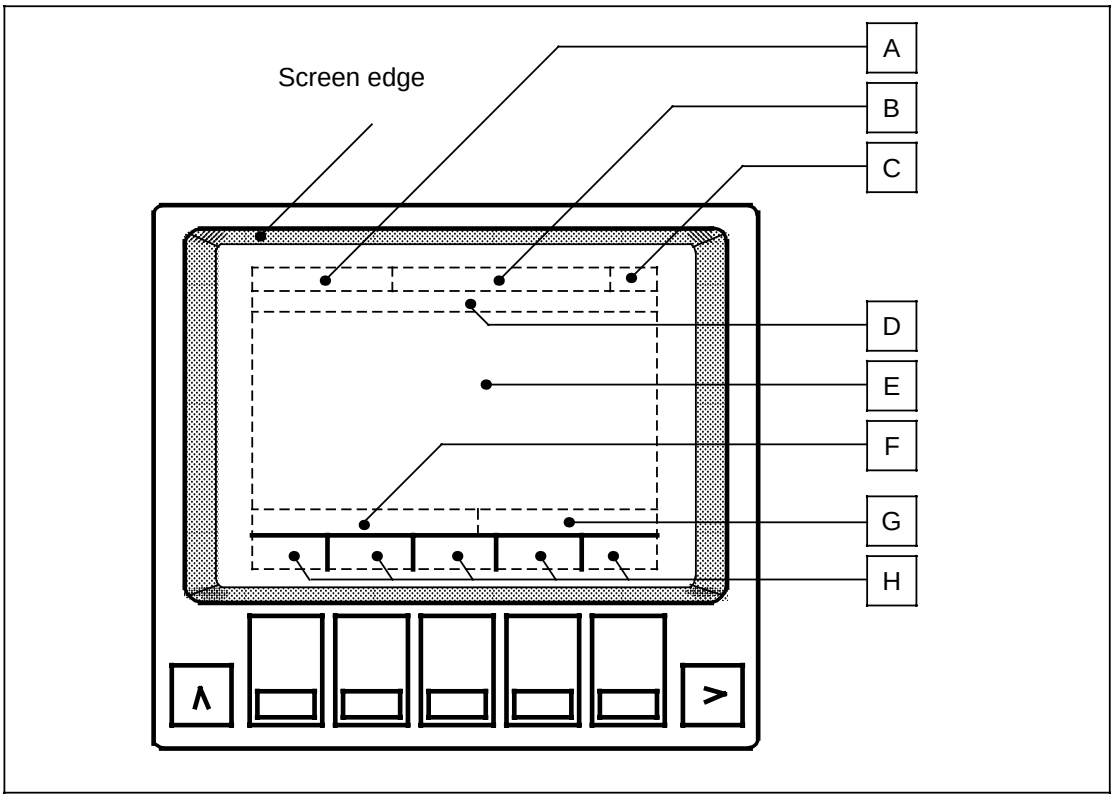


View of the SINUMERIK 810G GA3, operator interface with integrated machine control panel

Key to Fig. 2.1

1	CRT display with softkeys (see Section 2.1.1.1)	4	Editing and input keys (see Section 2.1.1.4)
2	Display panel (see Section 2.1.1.2)	5	Control keys (see Section 2.1.1.5)
3	Address/numeric keys (see Section 2.1.1.3)	6	Integrated machine control panel (see Section 2.1.1.6)

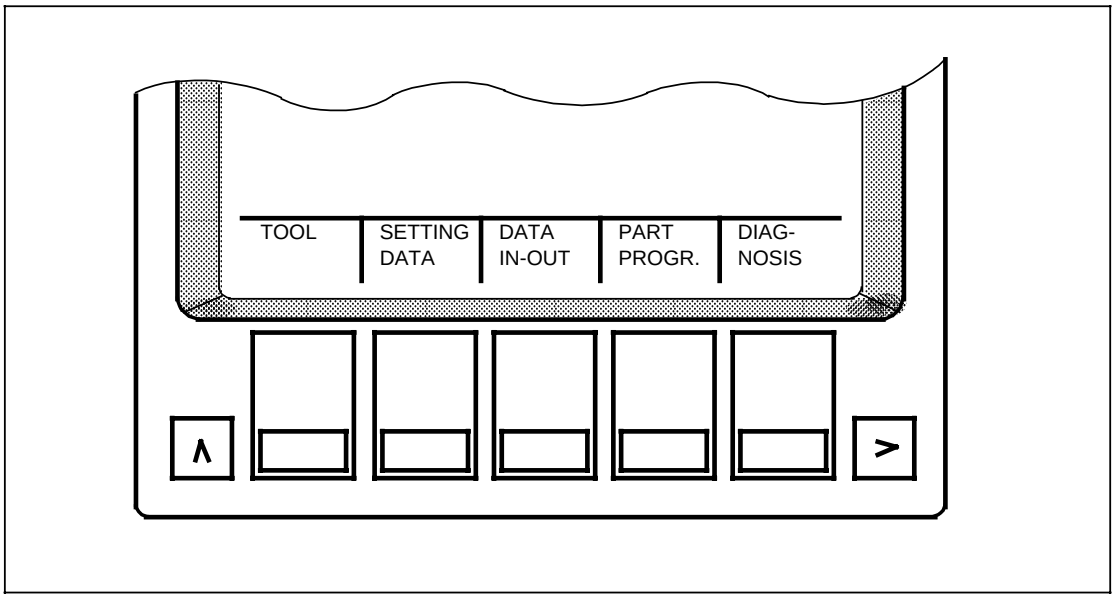
2.1.1.1 CRT display with softkeys



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

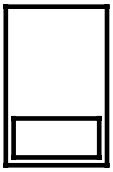
The CRT display is divided into **17 lines**, **each of 41 characters**. The following table shows the display function of the individual line / line area:

Note	C R T line	Display area for	Max. number of characters
A	1	Operating mode selected	14
B		Operating status	24
C		Channel number	3
D	2	Alarm No. and message text	41
E	3 to 14	NC displays: texts, graphics	41 x 12
F	15	Notes for the Operator	24
G		Inputs from keyboard	17
H	16 and 17	Softkey menu with up to five softkey functions	5 x 7 x 2



Keys below the CRT display

Softkeys



By pressing one of the five softkeys (a **softkey** is defined as a key that does not have a fixed function), you select the required softkey functions, shown in the menu directly above the softkey.

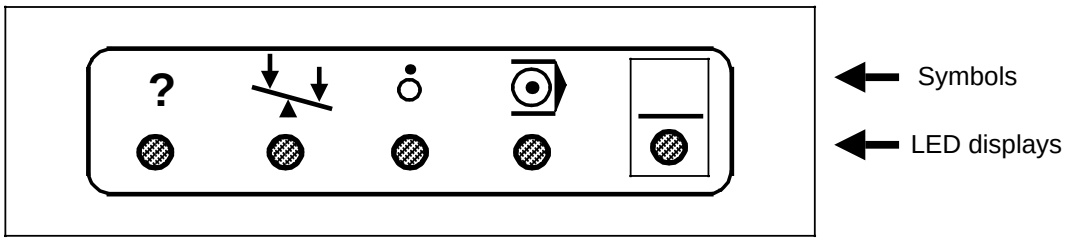
Key for extension of the same menu

By pressing this key, you change the displayed softkey functions in the text display.
Further functions for the **same menu** are shown.

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

By pressing this key, you change the softkey functions in the text display. You return to a **higher-level menu.**

2.1.1.2 Display panel



View of the display panel

"Alarm" LED

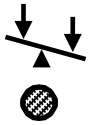
- The red LED lights up whenever there is a signal from the diagnostics monitor. The CRT display shows you the relevant message number and the message text (second line of display).
- The message numbers are explained in the alarm List in Section 4 of this Operator's Guide.
- The LED is extinguished when the message has been acknowledged.

with key  (Section 2.1.1.5)

with key  (Section 2.1.1.6 or 2.1.2.1).

Certain messages only disappear from the screen when the cause of the fault has been cleared (Section 4).

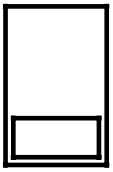
"Out of position" LED



- The green LED is lit when at least one axis is moving.
- The LED is extinguished, when all axes have reached their command positions.

If the  display does not go out disappear after a traverse motion, the drift has exceeded the permissible value. You **must** perform drift compensation.

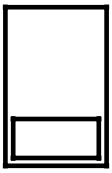
DIAGNOSIS



Press the "DIAGNOSIS" softkey.

Extend the softkey menu with this key located on the right-hand side below the screen.

NC MD



Press the "NC MD" softkey.

272* 

Call MD 272* with this key. Place the cursor on the required axis.

2720 1st axis
2721 2nd axis
2722 3rd axis
2723 4th axis
2724 5th axis

Press the "edit word" key.

The new compensation value is shown in the MD.

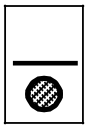
"Feed hold" LED

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

"Program running" LED

- The green LED is bright when a program is being executed, even if the machine is not moving.
- The LED is extinguished when program processing is completed:
  – with programmed stop M00, M01
  – at the end of a single block
  – at the end of the program.

”Key assignment” LED



- The yellow LED is bright:
  When the **lower** symbols of all the double-function keys on the address/numerical keyboard (2.1.1.3) are active:
  The **lower** character of the operated double-function key is shown in the input line (2.1.1.1).
- The yellow LED is extinguished:
  When the **upper** symbols of all the double-function keys on the Address/numerical keyboard are active:
  The **upper** character of the operated double-function key is shown in the input line.
- Both display states

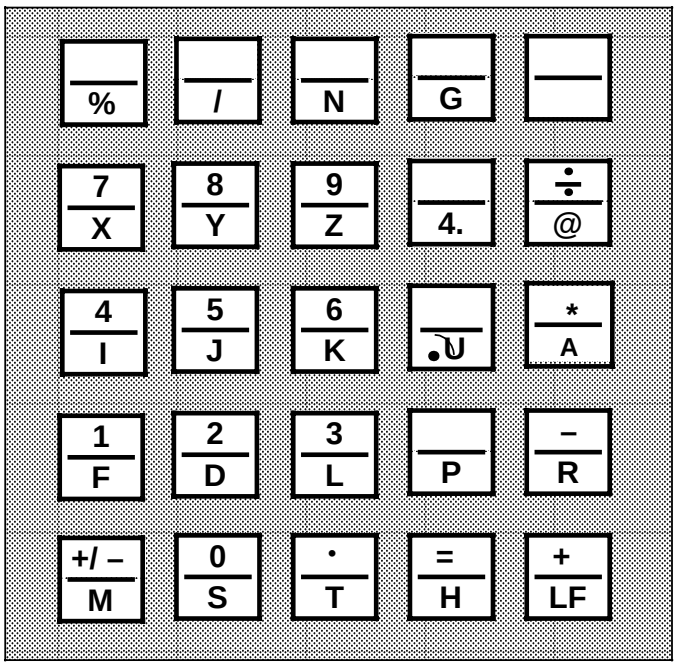
   ”yellow LED extinguished!”

   ”yellow LED lit!”

  are automatically switched over by the NC control.

- Using the key  on the address/numeric keyboard (2.1.1.3), you can also switch over ”manually”.

2.1.1.3 Address/numeric keys



View of the address/numeric keys

Description of the double function keys

%	Letter:	**a**[1]
	Symbol for:	**"Program start"**
/	Letter:	**b**[1]
	Symbol for:	**"Skip block"**
N	Letter:	**c**[1]
	Address for:	**"Block No." N ----**
G	Letter:	**d**[1]
	Address for:	**"Preparatory function" G ---**

1) *These letters are not permissible for normal programming. They are used for input of, or changes to, commands in "CL800" Machine Code (@...).*

Shift key:
Press this key to toggle between the upper or lower symbols of the double function keys.
- yellow LED (2.1.1.2) lit: lower symbols are active.
- yellow LED not lit: upper symbols are active.

Numeral: **7**
Address for: **"Position information" X ... Axis**

Numeral: **8**
Address for: **"Position information" Y ... Axis**

Numeral: **9**
Address for: **"Position information" Z ... Axis**

Letter **e**[1)]
Address for: **"Weginformation" Achse 4. .....**

Symbol for: **"Division"**
Address for: **"Program control function" @ ...**

Numeral: **4**
Address for: **"Interpolation parameter" I ...**
and: **"Infeed distance" I.....**

Numeral: **5**
Address for: **"Interpolation parameter" J .....**

Numeral: **6**
Address for: **"Interpolation parameter" K .....**
and: **"Infeed amount" K.....**

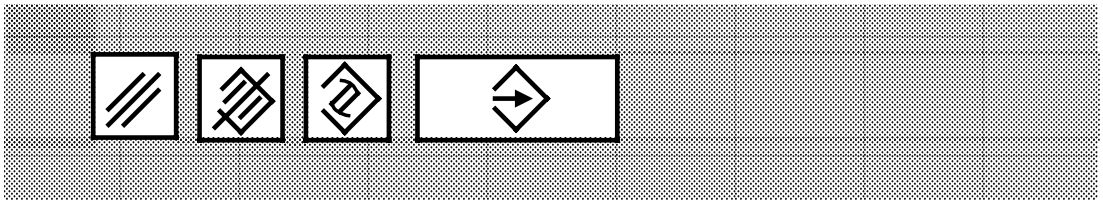
Letter **f**[1)]
Address for: **"Radius" U .....**

---

1) These letters are not permissible for normal programming. They are used for input of, or changes to, commands in "CL800" Machine Code (@...).

* / A	Symbol for:	**"Multiplication"**
	Address for:	**"Angle" A .....**
1 / F	Numeral:	**1**
	Address for:	**"Feed" F .....**
2 / D	Numeral:	**2**
	Address for:	**"Tool compensation number" D ...**
3 / L	Numeral:	**3**
	Address for:	**"Subroutine number" L ...**
~ / P	Symbol for:	(Tilde character)
	Address for:	**"Subroutine pass" P ..**
– / R	Symbol for:	**"Subtraction"**
	Address for:	**"Parameter " R ...**
+/ – / M	Sign changeover for	**+ –** or **– +**
	Address for:	**"Auxiliary function" M ..**
0 / S	Numeral:	**0**
	Address for:	**"Spindle speed" S .....**
• / T	Symbol for:	**"Decimal point"**
	Address for:	**"Tool number" T ....**
= / H	Symbol for:	**"Equals"**
	Address for:	**"Auxiliary function" H ....**
+ / LF	Symbol for	**"Addition"**
	Symbol for:	**"End of block" (Line Feed)**

2.1.1.4 Editing and input keys



View of the editing/input keys

Delet Input/ Operator Message

With this key, you delete:

- Characters in the input line (2.1.1.1)
  - press once to delete:
    the last character/character furthest right
  - keep pressed to delete:
    all characters consecutively, from right to left, until the input line is clear.
- Characters in the operator message line (2.1.1.1)
  - press once to delete:
    all characters simultaneously.

Delete word/block

With this key, you delete from the part program memory

- The **word** on the CRT display to the **right** of the cursor (2.1.1.5), if the **address** of this word is shown on the input line (2.1.1.1)
- The **block** on the CRT display to the **right** of the cursor; if the **number of this block** is shown in the input line.

Edit word

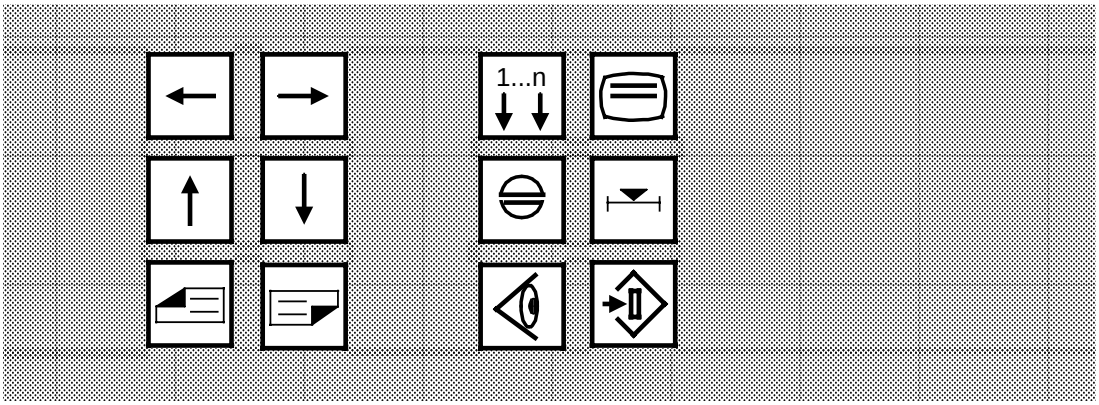
With this key, you edit the **word** on the CRT display to the **right** of the cursor in the part program memory (2.1.1.5):
The word marked with the cursor changes into the word with the **same address** that is shown on the input line (2.1.1.1).

Input character/word

You conclude your inputs with this key:

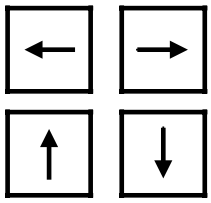
- The **characters** shown on the input line (2.1.1.1) are transferred into the **list display** or the **input field** on which the cursor (2.1.1.5) is positioned.
- A **word** shown in the input line is transferred into the **part program memory.**

2.1.1.5 Control keys



View of the control keys

Move cursor left/right
Move cursor up/down



With these keys you move the cursor on the CRT display.
SINUMERIK 810G differentiates between:

- Cursor for marking a word or block within displayed **part programs**
  The cursor looks like this: ▮
- Cursor for marking an input field within displayed **input forms**/ **dimensioned graphics**
  The cursor looks like this: ▬▬▬ or ▬
  (The length of the cursor depends on the max. permitted number of characters in the marked field).

Cursor movement in a **part program**

With this key, you move the cursor :

- from word to word, moving to the left
- to the end of the previous block (after the last word/in front of LF), if the cursor was positioned before the beginning of block.

With this key, you move the cursor :

- from word to word, moving to the right
- before the beginning of the following block, if the cursor was positioned at the end of a block (after the last word/in front of LF).

With this key, you move the cursor
- from the beginning of a block to the beginning of the previous block
- to the start of the same block if the cursor was positioned within a block
- to the start of program, if you keep the key pressed.

With this key you move the cursor
- from the beginning of a block to the beginning of the following block
- to the start of the following block if the cursor was positioned within a block
- to the end of program, if you keep the key pressed.

Cursor movement in an **input screen form / dimension graphic display**

With one of these keys, you move the cursor
- **Backwards** from field to field in an **input form**[1]: from right to left within a line and then from right to left in the line above etc.
- Backwards in a **dimensioned graphic display** to the previous point; if the key is operated continuously the cursor jumps to the input field for the first dimension to be input.

With one or the other of these keys, you move the cursor
- **Forwards** in an **input form**[1] from field to field: from left to right in the line, and then to the line below from left to right
- **Forwards** in a **dimensioned graphic display**, according to the previously entered sequence of dimension input. If the key is kept pressed, the cursor jumps to the input field for the last dimension to be input.

Paging up/down

By pressing one of these keys you can page through the current CRT displays.

Page one display **up.**

Page one display **down.**

---

1) Input forms are available for:
Tool compensations, setting data, machine data, operator guidance.

Channel changeover

The SINUMERIK 810G GA3 control has **three channels** :

- By pressing this key once, you switch to the next highest channel number to the one displayed in the top line of the display.
- Pressing the key again switches to the next highest channel etc.

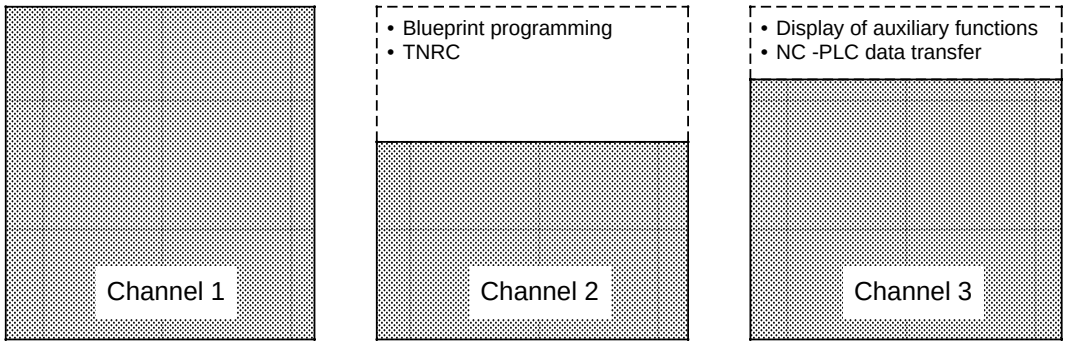
Explanation of the channel structure:

The three channels have the following significance:

Channel 1: **Main channel** for executing part programs and spindle programming.
Channel 2 : **Auxiliary channel** for executing programs for auxiliary axes or calculation functions in the background. The channel No. is displayed blinking and inverse.
Channel 3 : **Graphic simulation** for program display on the CRT display.
(Note: ”GRAPHIC SIMULATION” function is an Option!).

The use of channels makes it possible not only to edit programs and service interfaces while a part program is running but also to run two part programs at the same time.
All three channels can generally be operated simultaneously. There are, however, a few functions which may cause collision problems .

Function implemented in the channels (•...function not implemented):



The auxiliary channel (channel 2) is a fully functioning channel (with only a few functions not implemented. It is used mainly for calculations in the background and for auxiliary movements (such as tool change etc.).

The same axis can be moved in channels 1 and 2 if it is ensured that a traverse command cannot be output **simultaneously** in channel 1 and channel 2 (...alarm 180*: "axis programmed in both channels").

The auxiliary channel is mainly used to operate loader axes under PLC control simultaneously with the main channel.
Within the restrictions mentioned above, the auxiliary channel can be used for other purposes, too, matching it extremely flexible.
However, as it is only possible to transfer only M functions to the PLC from channel 2, the data exchange with the PLC is restricted.

See machine tool manufacturer's documentation:

- ***Is channel changeover used?***
- ***Which axes are assigned to which channel ?***

Edit input line

This key gives other keys an alternative function. It is pressed simultaneously with the other key. It is termed ALT (alternate) in the following text and is marked with a double line symbolizing an NC block in the part program.

The following functions are implemented:

ALT-EDIT	Copy NC block into input line
ALT-INPUT	Insert the contents into the input line without deleting
ALT-CLEAR	Delete entire input line
ALT-Cursor	Switch on cursor for overwriting the input line
ALT-Cursor	Switch off cursor for standard editing

Note:

- The NC block can only be up to 120 characters long.
- The corresponding setting data must be set (SD 5002 bit 6=1).
- The input line is limited to 16 characters.

Acknowledge alarm

By pressing this key:

- you **acknowledge** the information from the NC diagnostics displayed in the second line of the CRT display (2.1.1.1)
  - Error message text
  - Error message number

  for message Nos. 3000 3197
  and Nos. 6000 .... 6163.
  Program execution is not interrupted!
- you **clear** the red fault LED ? on the display field (2.1.1.2).

Actual position in double-height characters

When you press this key, the CRT display of ”Actual position” of the X, Y, Z axes (and fourth and fifth axes if present), is shown in double-height characters.

aus 0HQ01, S. 2-15
Änderungen a. Anlage

The ”distance to go” display remains in normal character size and including the path still to be travelled.
The CRT information previously displayed is cleared from the screen.
Pressing the key again will take you back to the previous display (with normal character size).

Diagnostics and installation

This switch is intended for:

- Installation
- Service

Please see the Installation Guide for further information.

Search for address, block no. or word or call up data

Press this key to search a **part program** for:

- an address
- a block number
- a word

or **display on the screen:**

- a tool compensation number (with the appropriate data)
- a machine data (MD) or setting data (SD).

Before pressing the search key you must write the data to be searched for into the input line (2.1.1.1).
Then you press the key and the cursor (looking like this: ▮)
jumps directly in front of the data searched for.

Please note when searching a part program:

- Addresses (other than the N... address) and words are only searched for from the current cursor position towards the end of program.
- If the data sought is between the start of the program and the current cursor position, it will not be found; the ”CHARACTER NOT FOUND” message is shown in the operator message line.
- Block numbers (e.g. ”N85”) are searched for and marked in both the directions, towards the program end and the program start.

2.1.1.6 Integrated machine control panel

Foto aus 0AJ02, S.2-15

View of the integrated machine control panel/SINUMERIK 810G GA3

Description of the keys

Reset

When you press the ”Reset” key:

- Execution of the current part program is aborted.
- Diagnostics messages are cleared
  (Alarm Nos.100 to 2999)
- The control is switched to the ”RESET” state:
  – the NC remains synchronized with the machine.
  – all buffer and working memories are cleared. (The part program memory remains unchanged).
  – the control is in the reset condition and ready for a new program start.

Single block

This key enables you to run a part program block by block in the ”AUTOMATIC ” operating mode.
When you press this key, the ”SBL” (single block) message is displayed in the first line of the CRT (2.1.1.1).

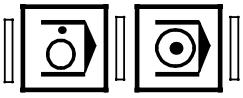
The current part program block is executed when you press the ”program start” key:

When the block has been executed, the message ”HOLD SINGLE BLOCK” is displayed on the CRT.
When you press the ”Program start” key again, the next block is called and executed etc.

You terminate single block mode by pressing the

key again.

Program stop / Program start (NC stop/NC start)



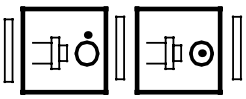
You press the ”program stop” key:

Execution of the part program is interrupted.
You can continue execution by pressing ”program start”.

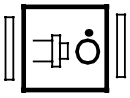
You press the ”program start” key:

The active part program is re-started at the current block. In Automaticmode , the overstored functions are transferred to the PLC.

Spindle stop/Spindle start



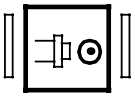
When you press the ”spindle stop” key:



- execution of the part program is stopped.
- the spindle is brought to a standstill.

Example of the use of ”spindle stop”

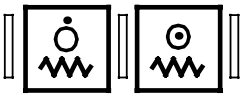
- an error is detected during execution of a block in ”MDI AUTOMATIC” mode.
- in ”JOG”, ”INC..”,”REPOS” modes (e.g. during repositioning to the contour).
- to change a tool
- to input S, T, H, M functions during setting up (overstore).

When you press the ”spindle start” key: 

- the part program continues in the current block
- the spindle runs at the speed specified in the part program.

The following values are set in machine data:
- ***the max. spindle speed***
- ***the values for the spindle speed override switch positions.***

Feed hold/Feed start



When you press the ”feed hold” key: 

- the program being executed is stopped.
- the feed drives are brought to a defined standstill.
- the red ”feed hold” LED lights up.

Examples of the use of ”feed hold”:

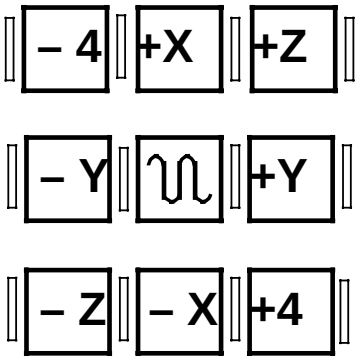
- an error is detected during execution of a block in MDI AUTOMATIC mode
- in ”JOG”, ”INC..”,”REPOS” modes (e.g. during repositioning to the contour)
- to change a tool
- to input S, T, H, M functions during setting up (overstore).

When you press the ”feed start” key: 

- the part program continues in the current block
- the feedrate reaches the value specified in the part program.

The following values are set in machine data:
- ***the feed and rapid traverse rates***
- ***the values for the feedrate override switch positions***
- ***whether the feedrate override switch is also active for rapid traverse.***

Direction keys/Jogging axes



This keypad enables you to jog the axes in the **”JOG”**, **”REPOS”** or ”**INC...**”, modes.

- The ”feed hold” LED must not be bright. 
- The CRT shows you the **specified** feedrate value ”F” at which the axes will travel when you press the direction key(s). The value is displayed as an absolute value, and as a percentage of the **programmed** feedrate (see: ”Feedrate decrease/increase” in this Section).
- You can traverse up to two axes simultaneously.
- In **”JOG”** the feed motion is controlled by the operator. The traverse path is only limited by the end limit switch.
- In **”REPOS”** the feed motion is controlled by the operator (see ”JOG”).
  If, however, the point of interruption (in a part program that has been partially run) is reached first, the direction keys become **inactive**.
- The direction keys can traverse the axes:
  – in **incremental mode**
  – in **continuous mode.**

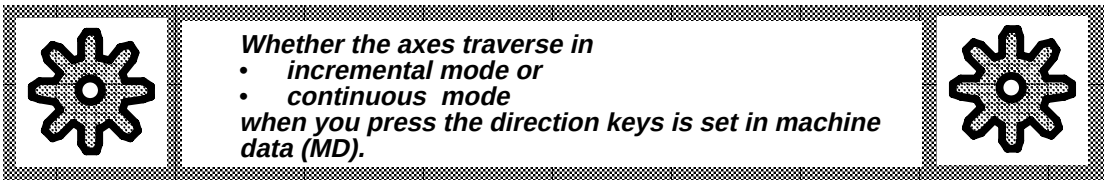
There are no keys on the integrated operator panel for the fifth axis.

Incremental mode:

When you press the direction key (however long you keep it pressed), the axis only traverses by one increment (set at 1**/**10**/**100**/** 1000**/**10000 µm).

Continuous mode:

The axis traverses as long as you keep the direction key pressed. Traversing stops when you release the key. This also happens if the set increment has not been reached.



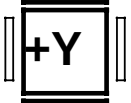
Whether the axes traverse in
- ***incremental mode or***
- ***continuous mode***

when you press the direction keys is set in machine data (MD).

Function of the direction keys

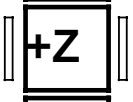
With this key you traverse the **"X"** axis.

With this key you traverse the **"X"** axis in the opposite direction.



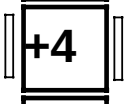
With this key you traverse the **"Y"** axis.

With this key you traverse the **"Y"** axis in the opposite direction.



With this key you traverse the **"Z"** axis.

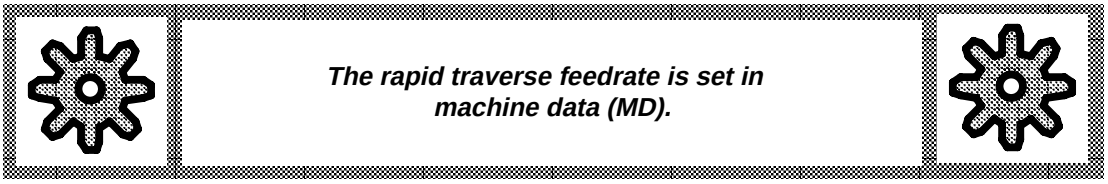
With this key you traverse the **"Z"** axis in the opposite direction.



With this key you traverse the **4th** axis (auxiliary axis).

With this key you traverse the **4th** axis (auxiliary axis) in the opposite direction.

Rapid traverse override:
When you press this key at the **same time** as any of the keys above, the axis is traversed at **rapid** traverse feedrate.



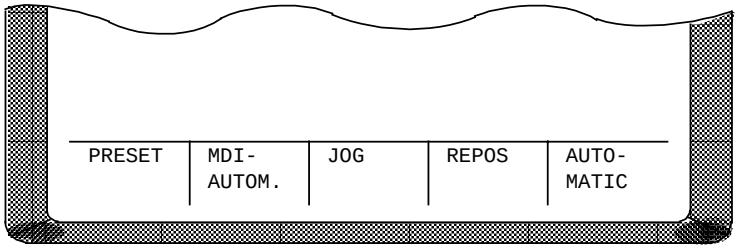
The rapid traverse feedrate is set in machine data (MD).

Key for the selection of operating modes

You use this key when you want to select operating modes or further softkey functions.
The menu selected is shown on the two bottom lines of the CRT display (menu for softkey functions (2.1.1.1)) .

Press the  **key**

until you see the menu for the following operating modes displayed:



Section of the CRT display with menu of the operating modes

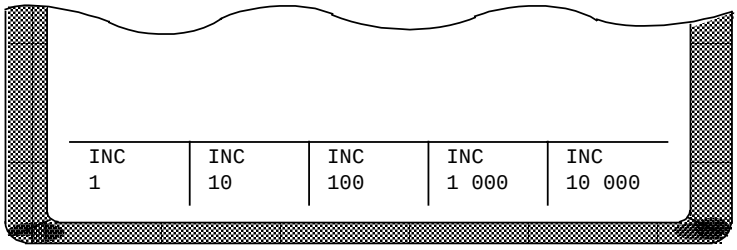
Press the  **key a second time.**

A menu of further softkey functions is displayed.
The menu displayed depends on the current operating mode, which is shown on the first line of the display (2.1.1.1).
(For detailed information see Section 2.3.3 to 2.3.6 and 2.4!)

You can extend the displayed operating mode menu (figure 2.9):

Press the key  **below the CRT.**

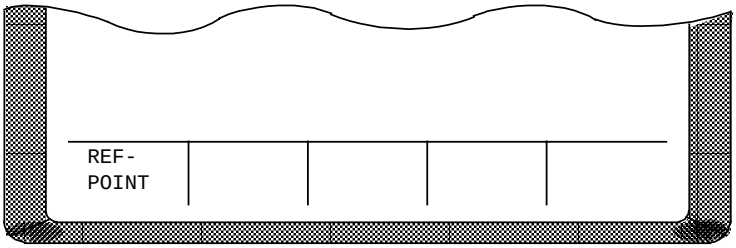
The extension of the operating mode menu is shown:



Section of the CRT display:
1st extension of the operating mode menu

Press the **>** **key a second time.**

A 2nd extension of the operating mode menu is shown.

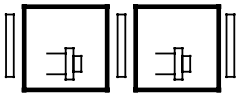


Section of the CRT display:
2nd extension of the operating mode menu

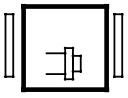
Press the **>** **key a third time**

The text display returns to the menu first selected according to figure 2.9 etc.

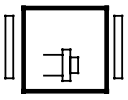
Spindle speed override decrease / increase



The two keys enable you to decrease or increase the **programmed** spindle speed value ”S” (with reference to 100% value).
The fixed spindle speed value ”S” set with these keys is displayed as an **absolute** value, and as a percentage.

Press the  **key briefly.**

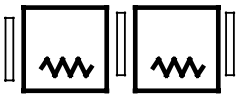
The spindle speed is **reduced by 5%**.
Keeping the key pressed causes the spindle speed to be **reduced** in steps of 5% until the final value of **0%** (standstill) is reached.

Press the  **key briefly.**

The spindle speed is **increased** by 5%.
Keeping the key pressed causes the spindle speed to be **increased** in steps of 5% until the final value of **120%** is reached.

The step values given here are valid for standard machine data (MD).
These data can be modified user-specifically by the machine tool manufacturer!

Feed or rapid traverse override decrease/increase



The two keys enable you to decrease or increase the **programmed** feedrate value ”F” (with reference to 100% value). The value set with these keys is displayed as an absolute value and as a percentage.

Press the  **key briefly**

The feedrate is **decreased** in the following steps:

- by 5 %, in the feed range 120 % to 70 %
- by 10 %, in the feed range 70 % to 10 %
- by 2 %, in the feed range 10 % to 2 %
- by 1 %, in the feed range 2 % to 0 %

When **kept pressed** the feedrate is **decremented** in steps until the final value of 0% (standstill) is reached.

Press the  **key briefly**

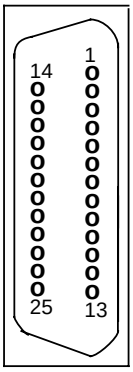
The feedrate is **increased** in the following steps:

- by 1 %, in the feed range 0 % to 2 %
- by 2 %, in the feed range 2 % to 10 %
- by 10 %, in the feed range 10 % to 70 %
- by 5 %, in the feed range 70 % to 120 %

When **kept pressed** the feedrate is **incremented** in steps until the final value of 120% is reached.

The step values given here are valid for standard machine data (MD).
These data can be modified user-specifically by the machine tool manufacturer!

Socket connector for universal interface



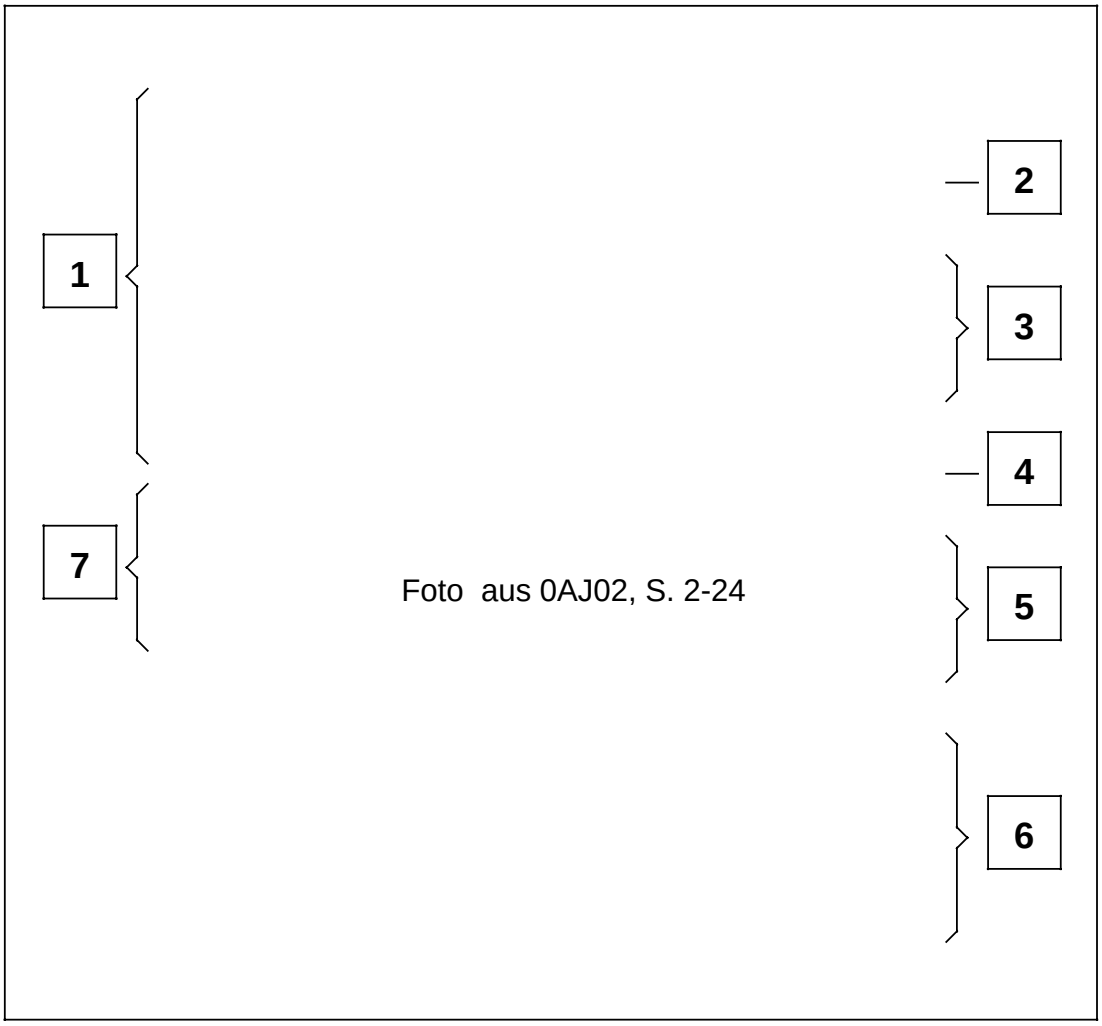
Beneath the cover hinged on the left, you will find a 25-pole socket for D-type subminiature connectors.

Via this RS232C (V.24) / 20mA interface, you can transfer serial data to or from peripheral devices.

For details of transmission data see Section 6.3, setting data, and for device connection see Section 6.4!

For details of the correct connecting cables see: "SINUMERIK System 800", Configuring Instructions/Universal Interface!

2.1.2 SINUMERIK 810G GA3 operator interface with external machine control panel



View of the SINUMERIK 810 G GA3 operator interface with external machine control panel

Key to Fig. 2.12

1	CRT display with softkeys (see Section 2.1.1.1)
2	Display panel (see Section 2.1.1.2)
3	Address / numerical keys (see Section 2.1.1.3)
4	Editing and input keys (see Section 2.1.1.4)
5	Control keys (see Section 2.1.1.5)
6	External machine control panel (see Section 2.1.2.1)
7	Operator panel with 24 freely assignable unlabelled keys, or blank panel with connector

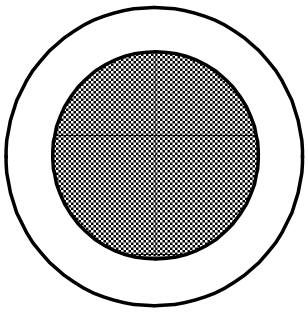
2.1.2.1 External machine control panel

Foto aus 0AJ02, S. 2-25

View of the external machine control panel

Explanation of the operating elements

Emergency stop switch



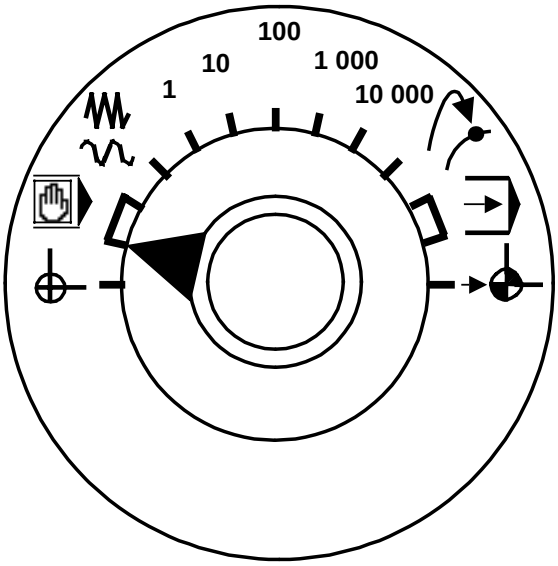
Press the red push-button in **emergency situations**:

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

Pressing of the ”Emergency stop” push-button generally brakes all drives with maximum braking power to a defined state and causes a ”RESET”.

Further or other reactions to ”Emergency stop” are possible

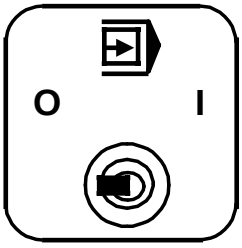
Operating mode selector switch



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

Symbol on the selector switch	Function	Designation of the operating mode
	**ACTUAL VALUE SETTING**	**PRESET** **Pre**set **Set**point (1st position)
	**MANUAL DATA INPUT/ AUTOMATIC**	**MDI AUTOMATIC** **M**anual **D**ata **I**nput - **Automatic** (2nd & 3rd position)
	**FEED/ JOG**	**JOG** **Jog**ging (4th position)
1, 10, 100, 1 000, 10 000	**INCREMENTAL TRAVERSE**	**INC FEED...** **Inc**remental **Feed** (5th to 9th position)
	**REPOSITIONING** Reapproach contour	**REPOS** **Repos**ition (10th position)
	**AUTOMATIC MODE** Execution of stored programs	**AUTOMATIC** (11th and 12th position)
	**TRAVERSE TO REFERENCE POINT**	**REFPOINT** **Ref**erence **Point** (13th position)

Single block switch



This key enables you to run a program block by block in the ”AUTOMATIC” operating mode.

When the switch is in position **”0”**: Single block mode is not active!

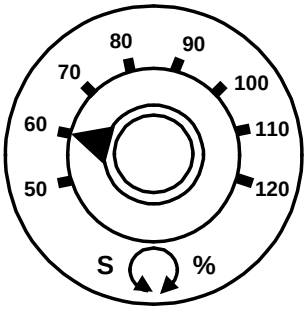
When the switch is in position **”I”**: Single block mode is active!

When single block mode is active:

- the ”SBL” (Single block) message is shown on the first line of the CRT (2.1.1.1)
- the current part program block is executed when you press the ”program start” key 
- when the block has been processed, message the ”HOLD SINGLE BLOCK” is displayed on the CRT
- when you press the ”program start” key again, the next block is transferred and executed etc.

The control inserts additional blocks for certain functions (e.g. "Coordinate rotation", "Soft approach to contour"). Depending on the number of blocks inserted, the "program start" key must be pressed several times.

Spindle speed override switch



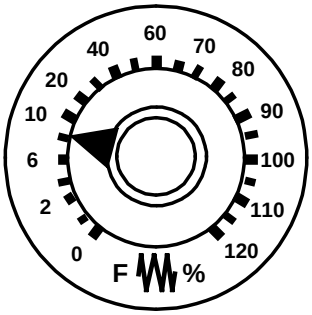
- The rotary switch, with 16 detent positions, enables you to decrease or increase the programmed spindle speed ”S” (relative to 100%).
- The actual function of the switch depends on a machine data.
- The set spindle speed value ”S” is displayed on the CRT as an absolute value and as a percentage.

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

Step value: 5% from detent position to detent position

The step value and control range given here apply to standard machine data (MD). These data can be modified user-specifically by the machine tool manufacturer!

Feed / rapid override switch



The rotary switch, with 23 detent positions, enables you to decrease or increase the **programmed** feedrate value ”F” (taken to be 100%).

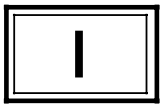
The set feedrate override value ”F” is displayed on the CRT in %.

Control range: 0% to 120% of the programmed feed rate.
In **rapid**, 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%

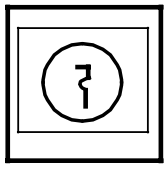
The step value and control range are given here valid for standard machine data (MD). These data can be modified user-specifically by the machine tool manufacturer.

Switch for switching on the NC control

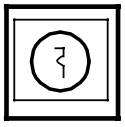


Press this key to switch the NC control on.

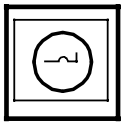
Keyswitch for input disable / operation disable



You can disable data input with the keyswitch
The operating functions affected are then no longer possible.
(e.g. COPY, DELETE, REORG, RENAME, MOVE)



The key can be withdrawn.



The key **cannot** be withdrawn.

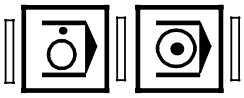
Whether or not disable is active in your control is set in a machine data.

Reset

You press the ”reset” key:

- The current part program is aborted.
- Diagnostics messages are cleared (Alarm No.100 to 2999).
- The control is switched to the ”Reset” state:
  – The NC remains synchronised with the machine.
  – All buffers and working memories are cleared (the part program memory remains unchanged).
  – The control is in the reset condition and ready for a new program start.

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



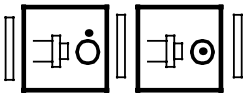
Press the ”program stop” key: 

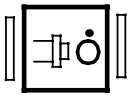
The program being executed is interrupted. You can continue execution by pressing ”program start”.

Press the ”program start” key: 

The part program called is re-started at the current block. In automatic mode the overstored functions are transferred to the PLC.

Spindle stop/Spindle start

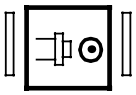


Press the ”spindle stop” key: 

- The program being executed is stopped.
- The spindle is brought to a standstill.

Example of the use of ”spindle stop”

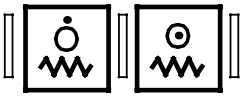
- An error is detected during execution of a block in ”MDI AUTOMATIC” mode.
- In ”JOG”, ”INC..”,”REPOS” modes; (e.g. during repositioning to the contour).
- To change a tool.
- To input S, T, H, M functions during setting up (overstore).

Press the ”spindle start” key: 

- The part program continues at the current block.
- The spindle speed accelerates to the value specified in the part program.

The following values are set in machine data:
- ***the max. spindle speed***
- ***the values for the spindle speed override switch positions***

Feed stop/Feed start



Press the ”feed stop” key: 

- The program being executed is stopped.
- The feed drives are brought to a standstill.
- The red ”Feed stop” LED lights up.

Example of the use of ”feed stop”

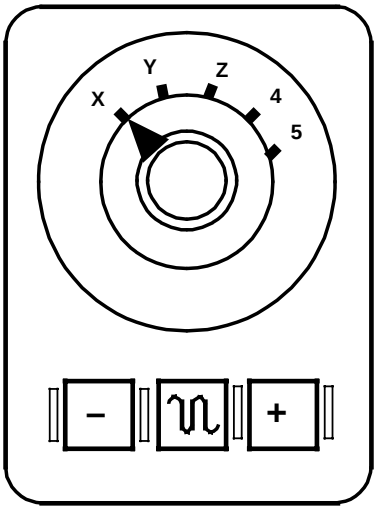
- An error is detected during execution of a block in ”MDI AUTOMATIC” mode.
- In ”JOG”, ”INC..”,”REPOS” modes, e.g. during repositioning to the contour.
- To change a tool.
- To input S, T, H, M functions during setting up (overstore).

Press the ”feed start” key: 

- The part program continues at the current block
- The feedrate is accelerated to the value given in the part program

The following values are specified in machine data:
- ***the feed and rapid traverse rates***
- ***the values for the feedrate override switch positions***
- ***whether the feedrate overrirde switch is also active for rapid traverse.***

Axis selector switch with direction keys/jogging axes



This keypad enables you to jog the axes in the **"JOG"**, **"REPOS"** or "**INC...**" modes.

- The "feed stop" LED must not be bright.

- The CRT shows you the **specified** feedrate value "F" at which the axes will travel when you press the direction key(s). The value is displayed as an absolute value and as a percentage of the **programmed** feedrate (see: "Feedrate decrease/increase", in this section).
- In **"JOG"** the feed motion is controlled by the operator. The traverse path is only limited by the end limit switch.
- In **"REPOS"** the feed motion is controlled by the operator (see "JOG").
  If, however, the point of interruption (in a part program that has been run) is reached first, the direction keys become **inactive**.
- The direction keys can traverse the axes:
  – in **incremental mode**
  – in **continuous mode**

Incremental mode:

When you press the direction key (however long you keep it pressed) the axis only traverses one increment (1/10/100/1000/10000 µm, depending on the setting).

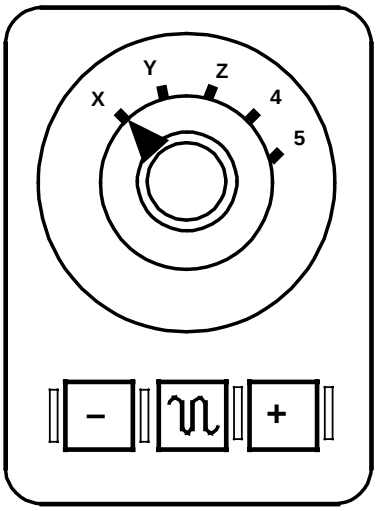
Continuous mode:

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

Whether you traverse
- ***incremental mode***
- ***continuous mode***

when the direction keys are pressed is set in machine data (MD)

Function of the axis selector switch with direction keys:

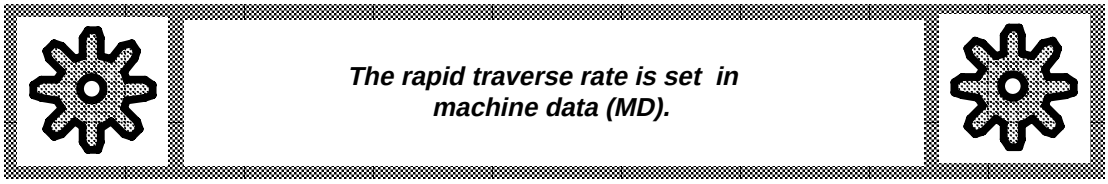


Select the axis to be traversed with the axis selector switch (X axis in the example).

Traverse the axis selected.

Traverse the axis selected in the opposite direction.

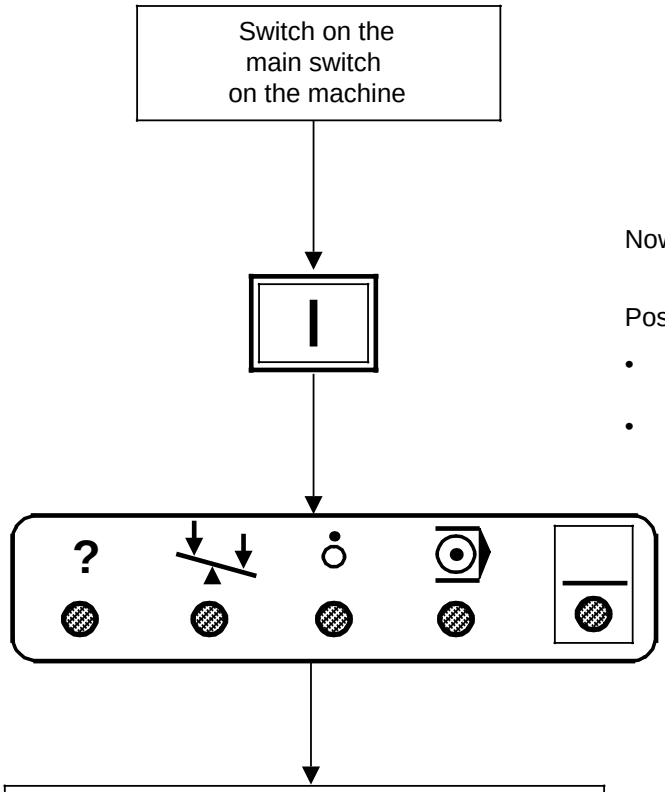
Rapid traverse:
When you press this key **together** with either of the above keys, the axis is traversed at **rapid**.



The rapid traverse rate is set in machine data (MD).

2.2 Switching on/off

2.2.1 Switching on the SINUMERIK 810G GA3



Now press the **"ON"** push-button!

Possible location of the push-button:

- External machine control panel (2.1.2.1)
- Machine operator panel.

Reaction of the display panel (2.1.1.2) on the control:

For approx. 3 seconds the 4 LEDs on the left are bright and then extinguished.

aus 0HQ01, S. 2-35
Änderungen s. Anlage

Approx. 10 seconds after switching on, you will see the basic display on the CRT, e.g. JOG.

The control is switched on!

2.2.2 Switching off the control

You switch the SINUMERIK 810G GA3 off with the main switch on the machine.

2.3 Operating modes

2.3.1 General notes

On a machine tool an NC controls according to instruction in a **part program**

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

In addition, preparations must be made on a numerically controlled machine tool before the actual machining process can be started.
For these preparations, the control has to be set to certain **operating states** to prepare it for certain operations.

These include:

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

The SINUMERIK 810 has **seven operating modes** for setting the control to the required state.

2.3.2 Overview of operating modes

The control has the following operating modes:

- **Automatic mode** (CRT Display: **AUTOMATIC**)

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

The part program can be entered into the control via the universal interface (e.g. via punched tape) or via the keyboard. While one part program is being processed, another part program can be entered.

aus 0HQ01, S. 2-36
Änderung s. Anlage

"AUTOMATIC" basic display

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

With the direction keys and the preset feedrate value ”F”, you can move the tool wherever you want.

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

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

aus 0HQ01, S. 2-37
oben
Änderung s. Anlage

”JOG” basic display

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

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

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

Application:

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

aus 0HQ01, S. 2-37
mitte
Änderung s. Anlage

”MDI AUTOMATIC” basic display

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

When the direction keys are used, the machine moves in either incremental or continuous mode, depending on the machine data set.

The reference point must be approached in each axis separately.

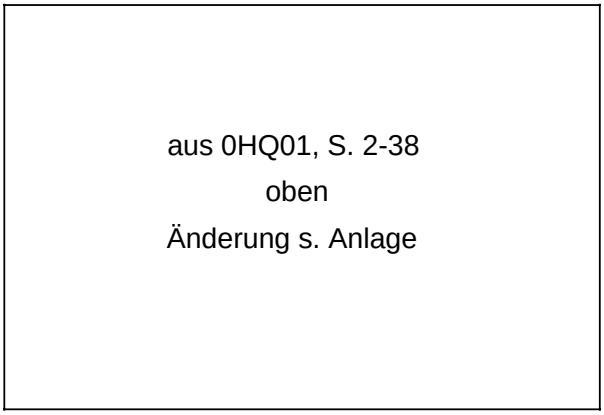
When the reference point has been reached, the position register is set to the value of the reference point coordinates.

aus 0HQ01, S. 2-37
unten
Änderung s. Anlage

”REFPOINT” basic display

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

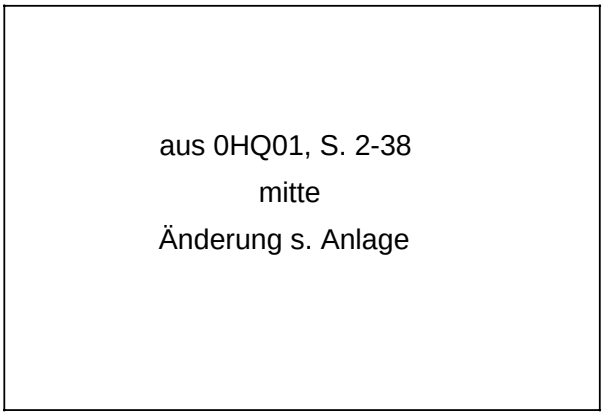
In this operating mode, defined paraxial positioning is possible using the direction keys. The feedrate is fixed with a machine data. Feedrate override (in the range 0%...120%) is only possible when the appropriate interface signal is transferred from the PLC to the NC.
MD define whether the set increment (in example shown: 100µm) will be traversed in incremental mode or continuous mode.



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

- **Actual value setting** (CRT display: **PRESET**)

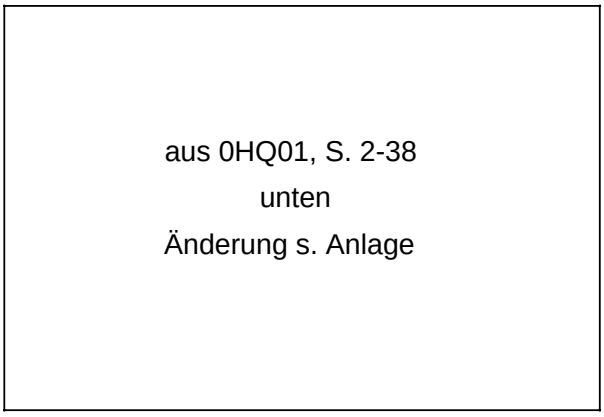
The directions of movement of an NC machine are based on a right-angled system of coordinates assigned to the individual machine axes.
In the absolute machine coordinate system, control zero can be shifted with reference to the machine zero point.
In "PRESET" mode, the control zero point can be placed anywhere within the machine coordinate system.



"PRESET" basic display

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

In "REPOS" mode, the tool can be returned to the point of interruption using the direction keys and the set feedrate value "F".
The "REPOS offset" display shows the distance from the actual position to the point of interruption, with the sign to show the direction of traverse.
When the point of interruption is reached, the "REPOS offset" display shows zero and the direction keys become inactive.



"REPOS" basic display

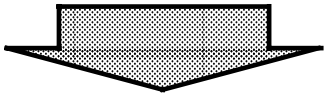
2.3.3 Selection of operating modes

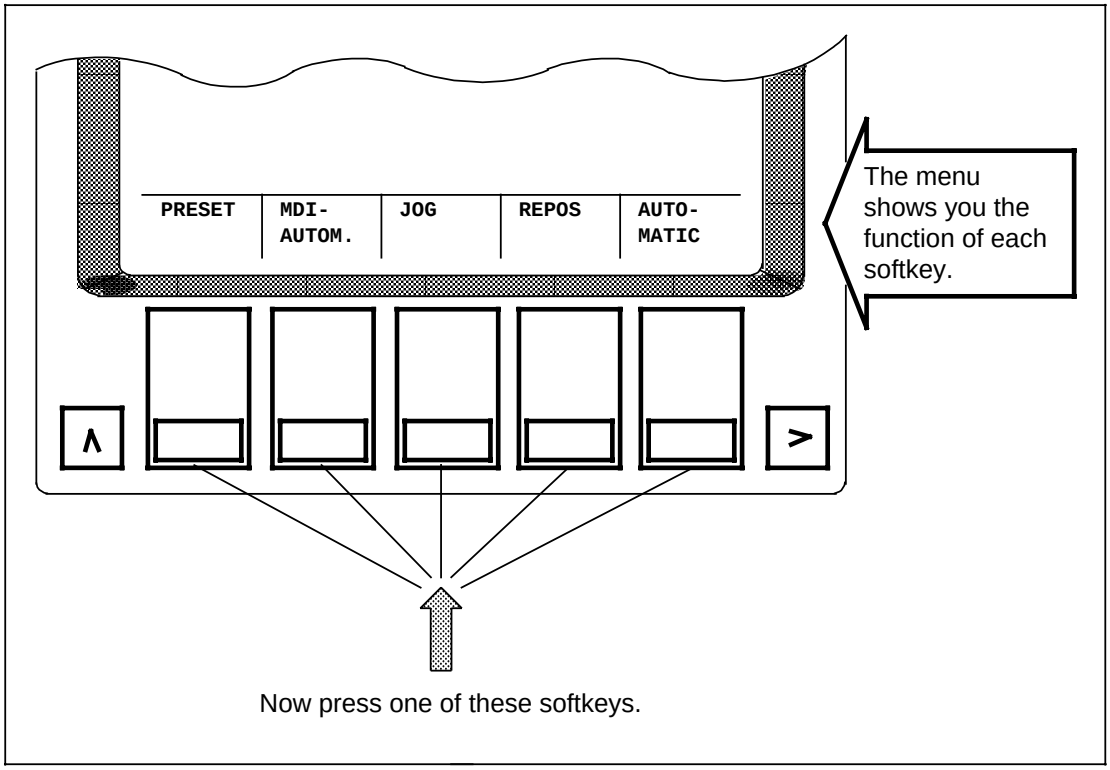
2.3.3.1 Selection of operating modes with integrated machine control panel

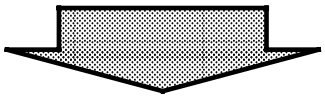
The following operating modes are selected as described below:

- PRESET (Actual Value Setting)
- MDI AUTOMATIC (Manual Data Input/Automatic)
- JOG (Feed/Jog)
- REPOS (Repositioning)
- AUTOMATIC (Automatic mode).

Press this key on the machine control panel.





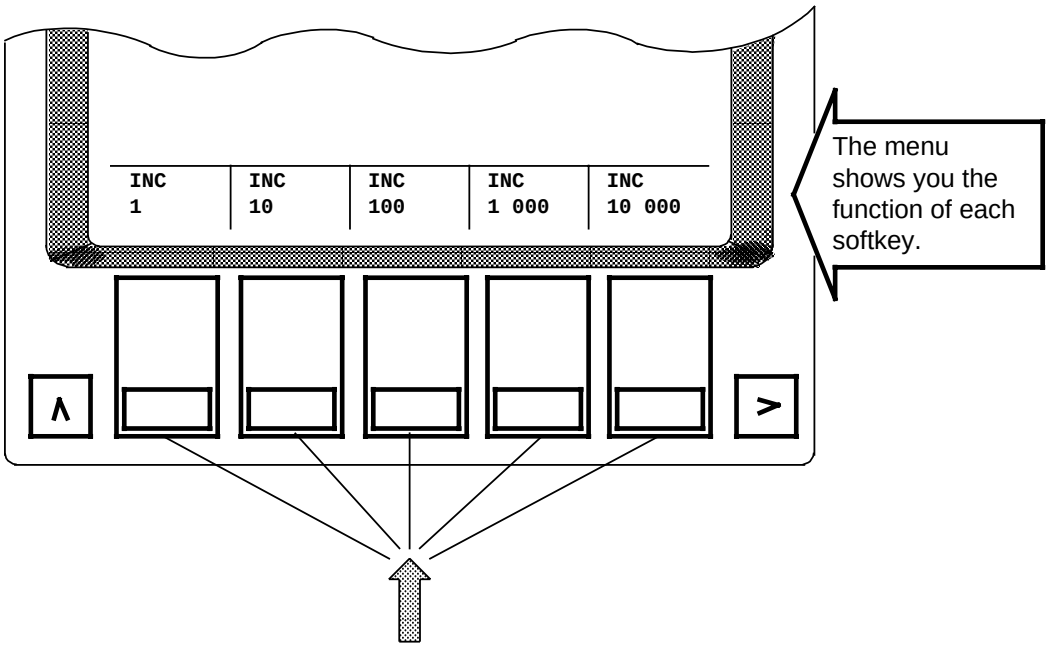


The basic display of the operating mode you selected is shown on the CRT
(See Figs. on page 2-36 to 2-38).

Further selection: Operating Mode **INC FEED 1 . . . . INC FEED 10 000 (incremental jog)**

Press the  **key on the machine control panel.**

Press the  **key under the CRT display once.**

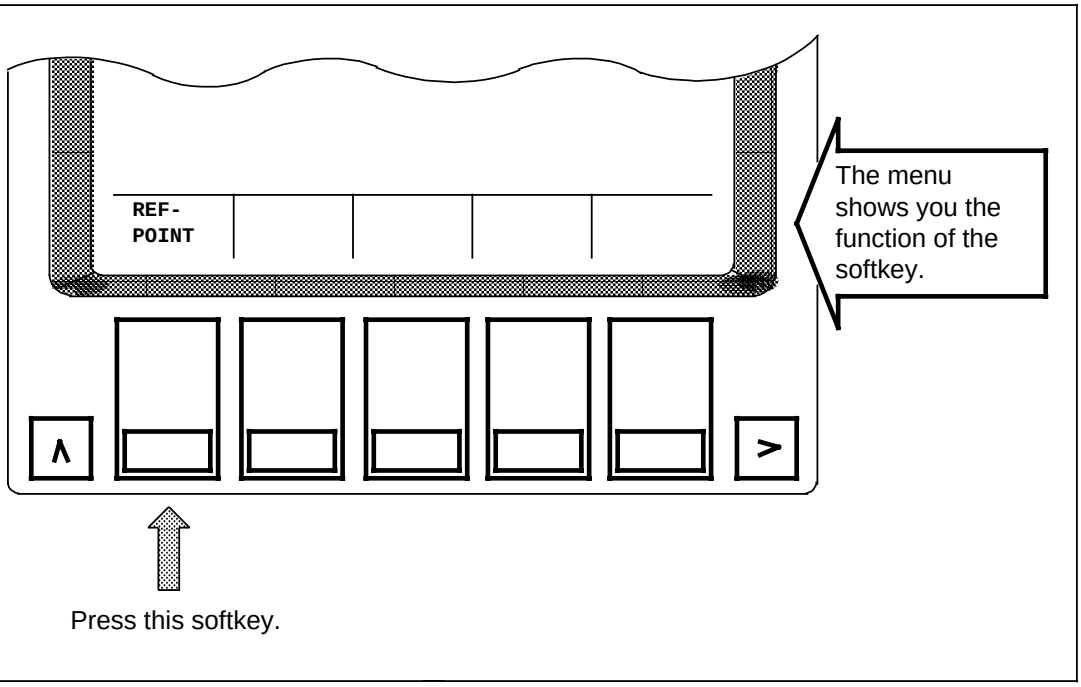


The basic display for ”INC 1.. INC 10000” mode is shown on the CRT display (Fig. on page 2-38).

Further selection: **REFPOINT mode (traverse to reference point)**

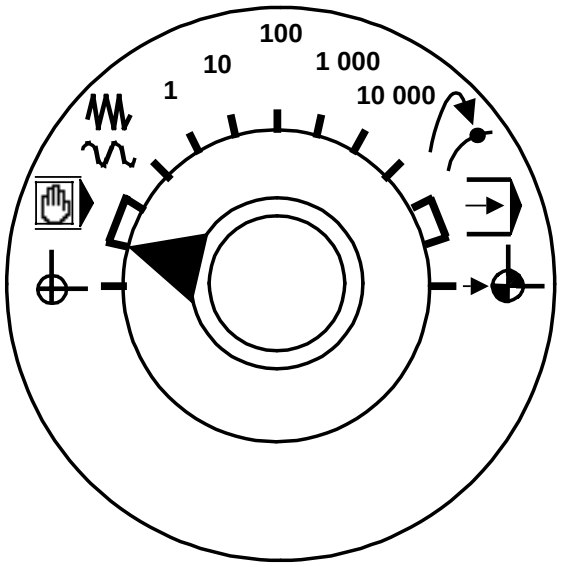
Press the  **key on the machine control panel.**

Press the **>** **key under the CRT display twice.**



The basic display for ”REFPOINT” mode is shown on the CRT display (Fig. on page 2-37).

2.3.3.2 Selection of operating modes with external machine control panel



Rotate the operating mode selector switch to the desired position.

Symbol on the selector switch	Function	Designation of the operating mode
	**ACTUAL VALUE SETTING**	**PRESET** **Pre**set **Set**point (1st position)
	**MANUAL DATA INPUT/ AUTOMATIC**	**MDI - AUTOMATIC** **M**anual **D**ata **I**nput - **Automatic** (2nd and 3rd position)
	**FEED/JOG**	**JOG** **Jog**ging (4th position)
1, 10, 100, 1 000, 10 000	**INCREMENTAL FEED**	**INC FEED...** **Incr**emental **Feed** (5th to 9th position)
	**REPOSITIONING** Reapproach contour	**REPOS** **Repos**ition (10th position)
	**AUTOMATIC MODE** Execution of stored programs	**AUTOMATIC** (11th and 12th position)
	**TRAVERSE TO REFERENCE POINT**	**REFPOINT** **Ref**erence **Point** (13th position)

2.3.4 "Reset" on change of operating mode

- When changing from one selected operating mode to another (Handling see 2.3.3.1 and/ or 2.3.3.2), a "RESET" can be generated by the control.
- The "RESET" generated by the control when changing mode has the same effect as if the "RESET" key had been pressed (see Sections 2.1.1.6 or 2.1.2.1).
- Whether a "RESET" is generated or not depends on which mode is changed to which mode (see Fig. 2.21).

	PRESET	MDI AUTOMA.	JOG	REPOS	AUTO-MATIC	INC ...	REF-POINT
PRESET		+	+	+	+	+	+
MDI AUTOMATIC	+		+	+	+	+	+
JOG	+	+		o	o	o	+
REPOS	+	+	o		o	o	+
AUTO-MATIC	+	+	o	o		o	+
INC ...	+	+	o	o	o		+
REF-POINT	+	+	+	+	+	+	

Generation of "RESET" with change of operating mode (+ ... reset, o ... no reset)

Explanations:

1. When changing from "AUTOMATIC" to "JOG" mode, **no** "RESET" is generated by the control!
2. When changing from "JOG" to "REFPOINT" mode, a "RESET" is generated by the control!

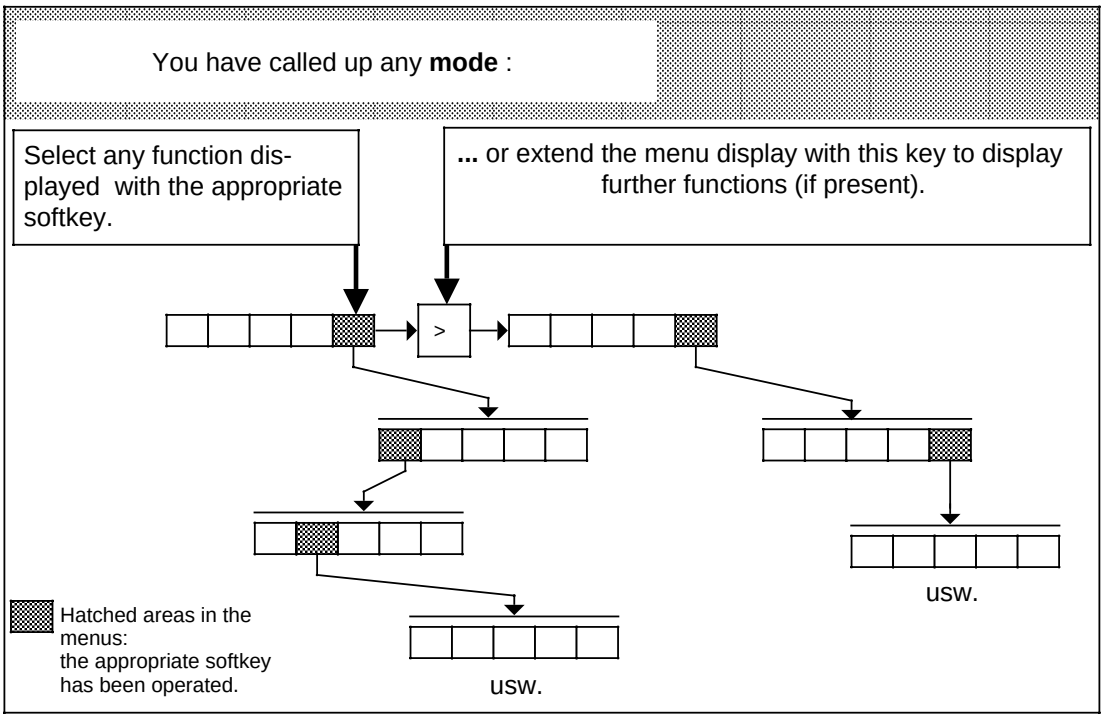
2.3.5 Branching to operating functions within an operating mode

Depending on the mode you have selected, a ”basic softkey menu” of operating functions is displayed in the **menu display** of the CRT (See Section 2.1.1.1).

After you have selected a function using one of the softkeys of this menu, the control will display other/new menus.

You can thus move through ”menu trees”with several branches.

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



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

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

2.3.5.1 Example of the selection of operating functions and branching to other menus

To select and branch to **other** menus, you simply use the 5 softkeys (see Section 2.1.1.1) on the CRT.

Example:

aus 0HQ01, S. 2-45
oben
Änderungen s. Anlage

Basic CRT display with the selected operating mode, e.g. ”AUTOMATIC”.
A menu of five selectable operating functions is shown.

You want, for example, to select the ”PART PROGRAM” function:
Press the appropriate softkey !

aus 0HQ01, S. 2-45
Mitte
Änderungen s. Anlage

The ”PART PROGRAM” function in ”AUTOMATIC” mode is shown on the CRT display.
A menu of five more selectable functions is shown.

You want, for example, to select the ”EDIT” function:
Press the appropriate softkey!

aus 0HQ01, S. 2-45
unten
Änderungen s. Anlage

The ”PART PROGRAM” menu in ”AUTOMATIC” mode is shown on the CRT display for the ”-EDIT” function selected.
The menu shows a branch to a new menu of three selectable operating functions.

usw.

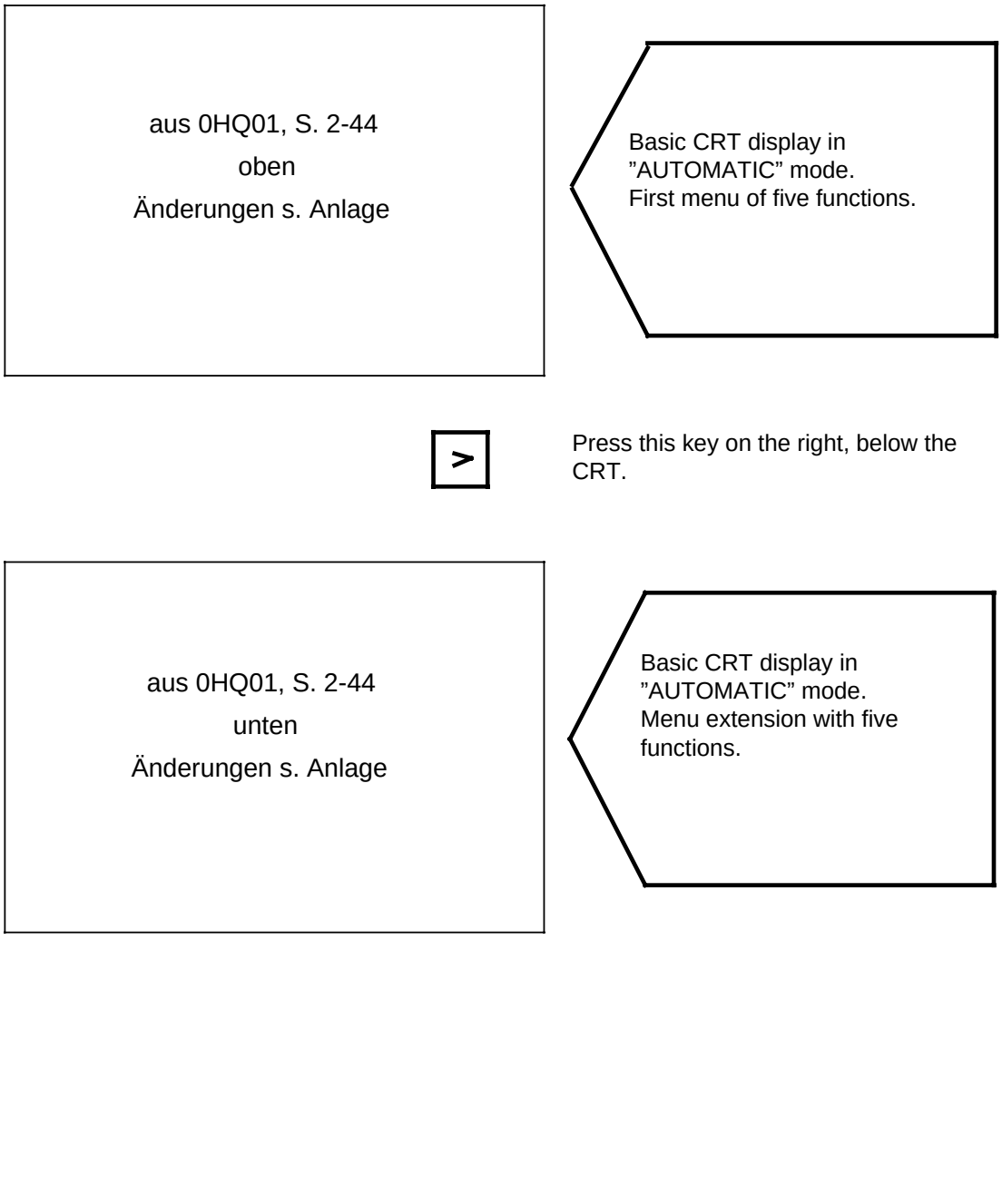
2.3.5.2 Example of the selection of further operating functions within the same menu

The menu on the CRT can display max. five functions.

To call further operating functions stored in the control **in the same menu**,

you use the key: [>]

Example:

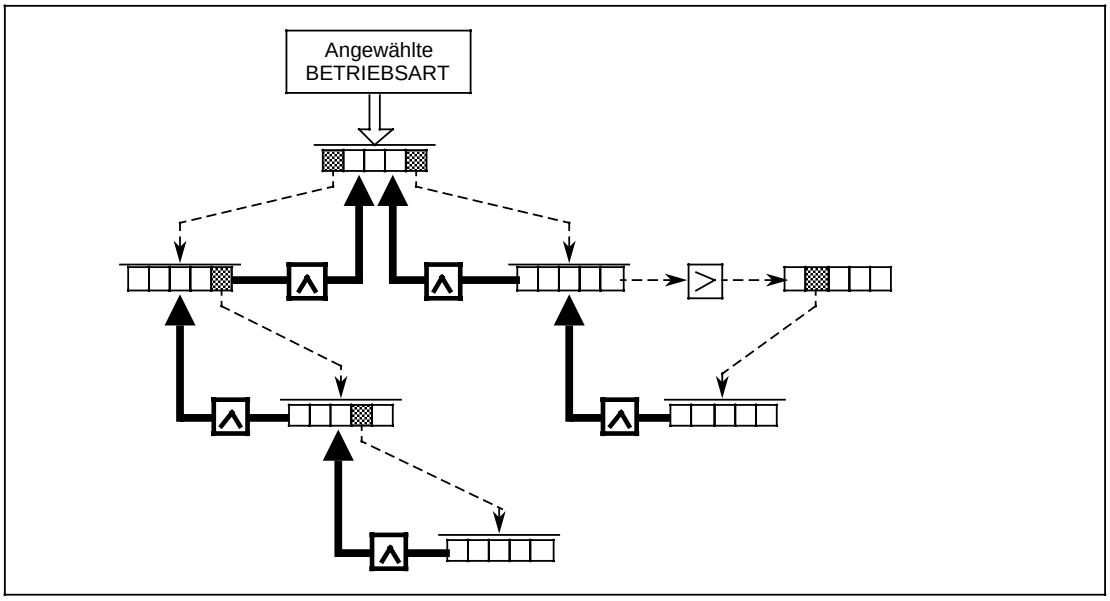


2.3.6 Jumping back to operating functions in higher-level menus within an operating mode

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

To do this, use the key: 

If you press this key once, the next highest-level menu with operating functions is displayed.



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

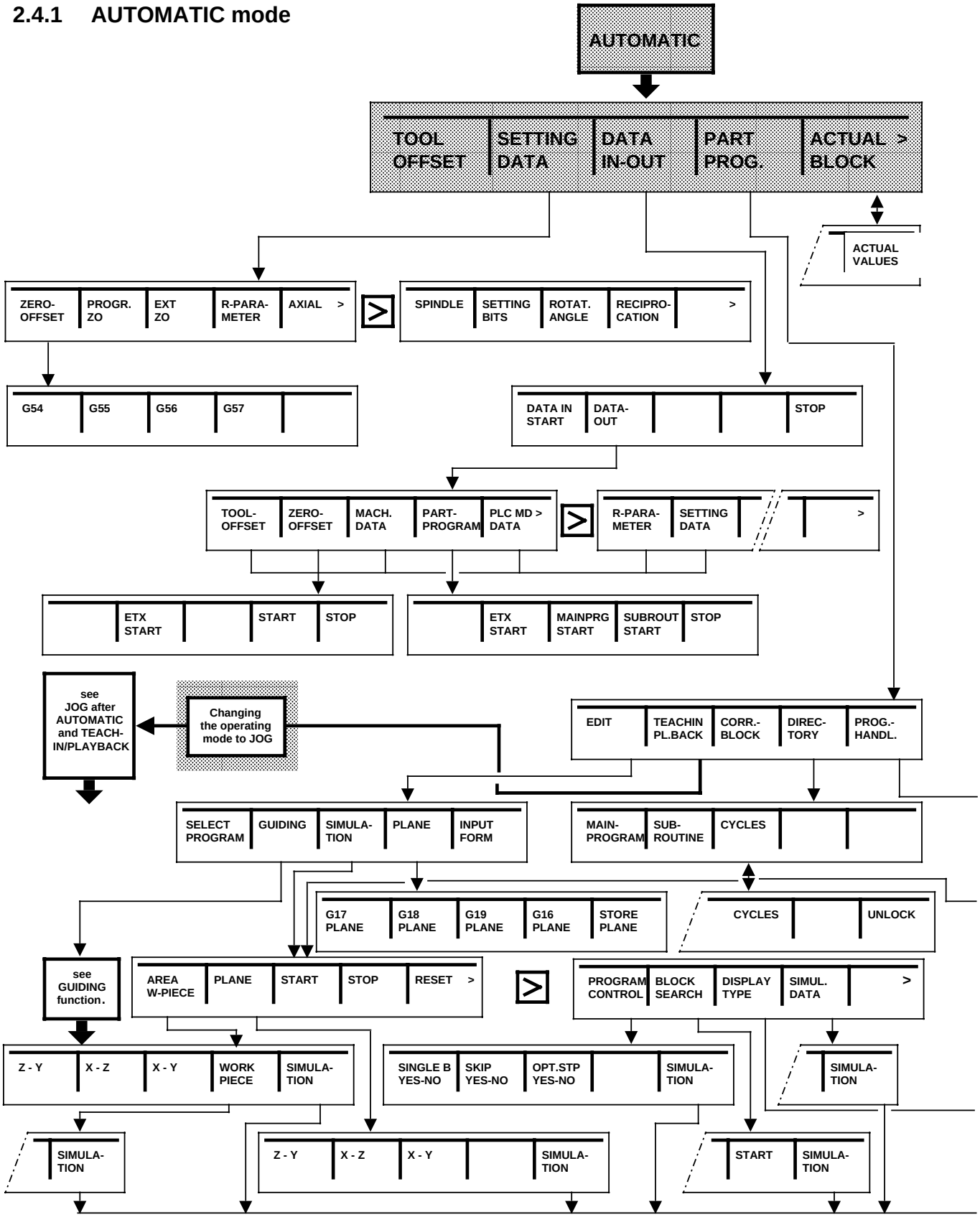
2.4 Operating mode menu trees

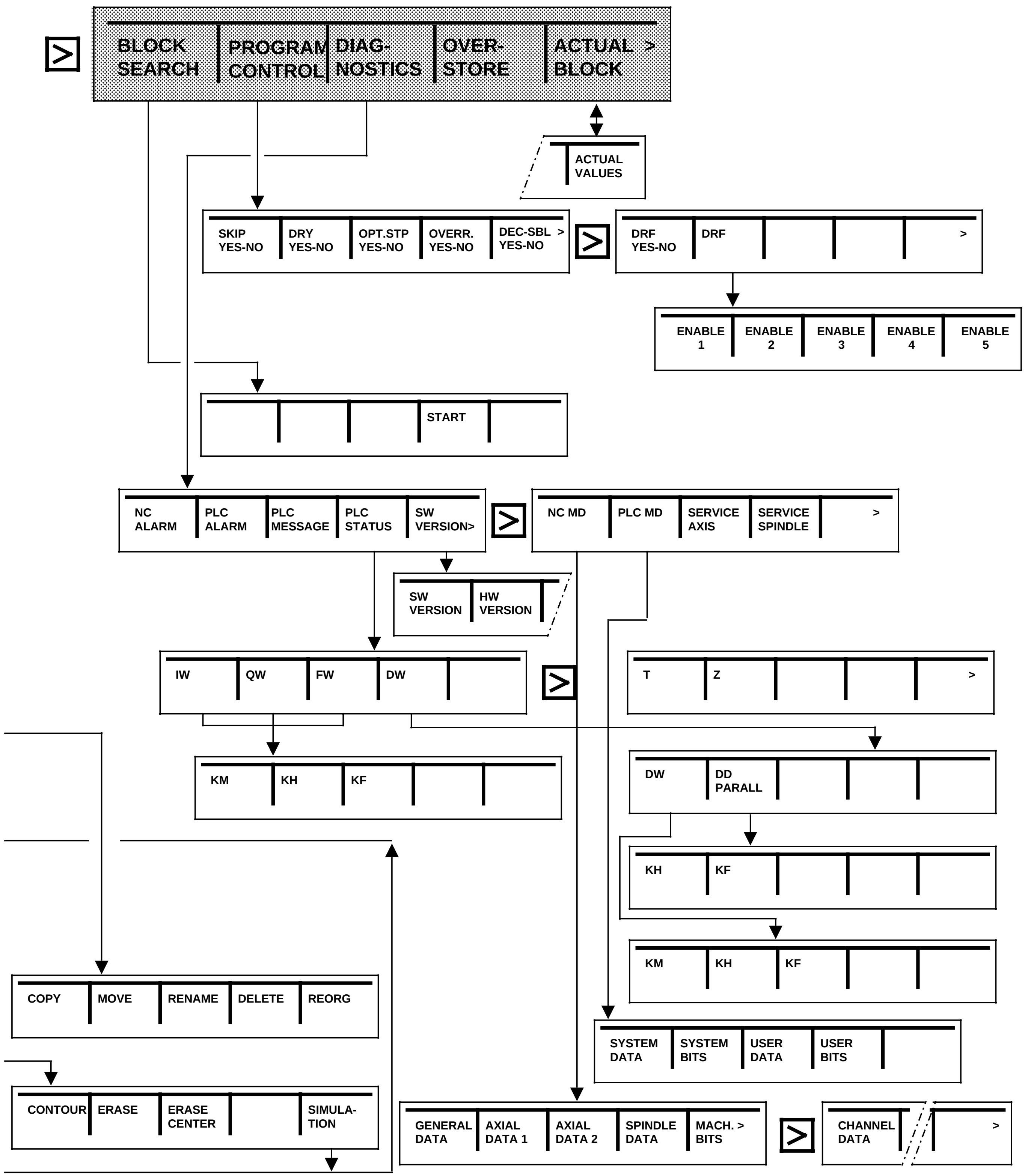
In this section you will find graphical overviews for **all** the SINUMERIK 810G **operating modes**.

All the branches to further functions are represented for:

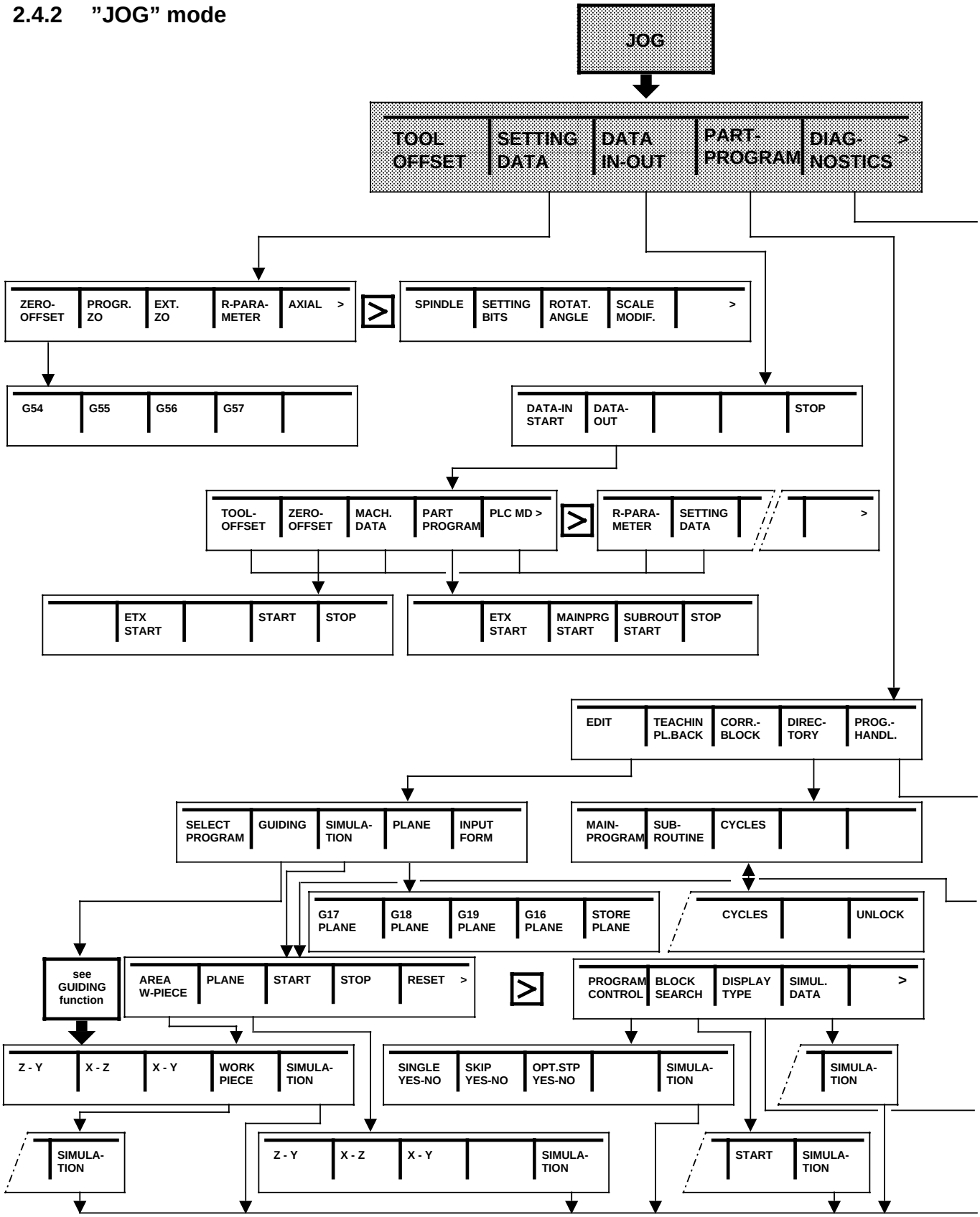
- "**AUTOMATIC"** mode
- "**JOG"** mode
- "**JOG"** mode (after "TEACH IN/PLAYBACK" function in AUTOMATIC mode)
- "**MDI AUTOMATIC"** mode
- "**REFPOINT"** mode
- "**INC 1 ... INC 10000"** mode
- "**PRESET"** mode
- "**REPOS"** mode
- **"GUIDING"** (operator prompting) function

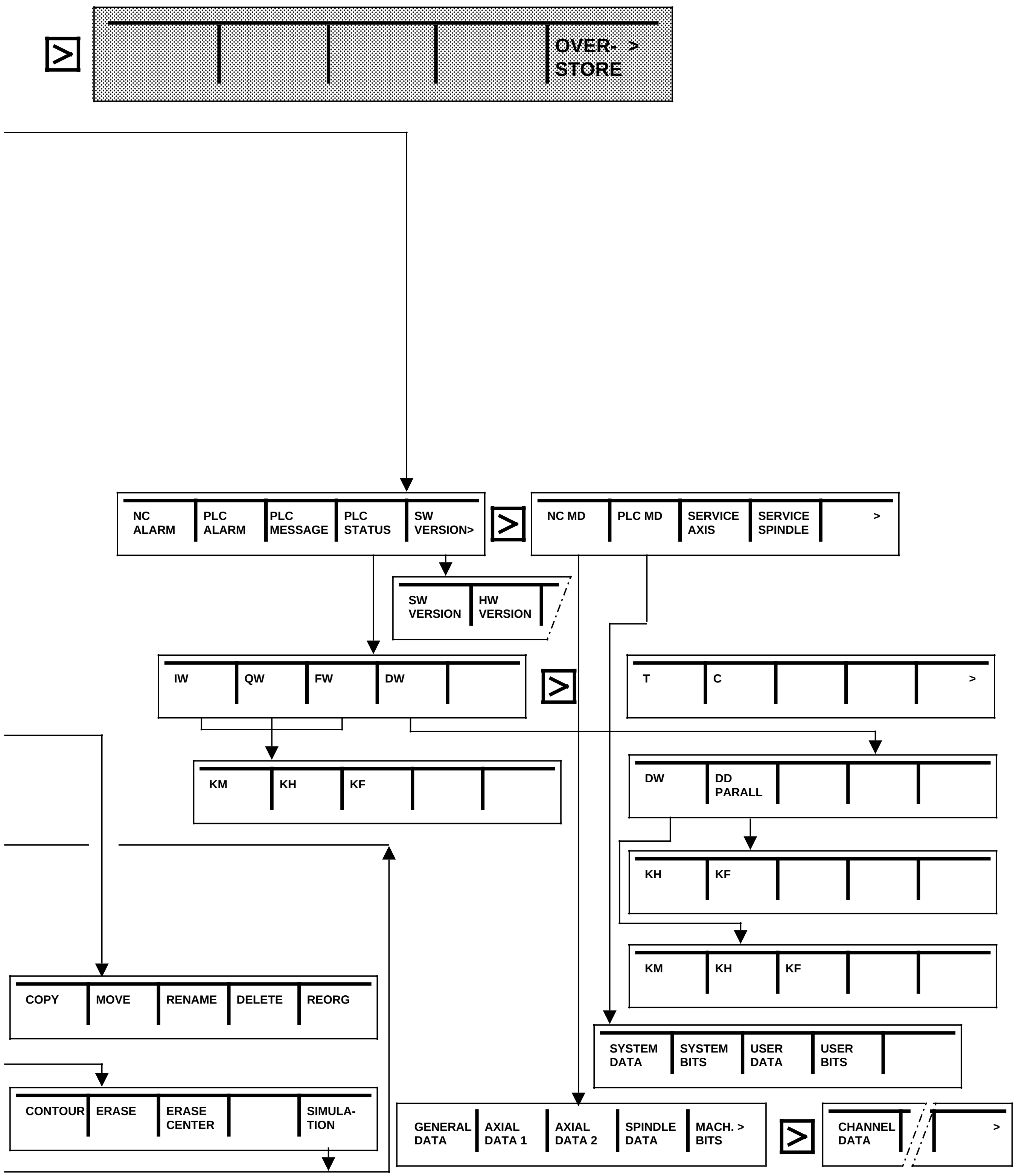
2.4.1 AUTOMATIC mode



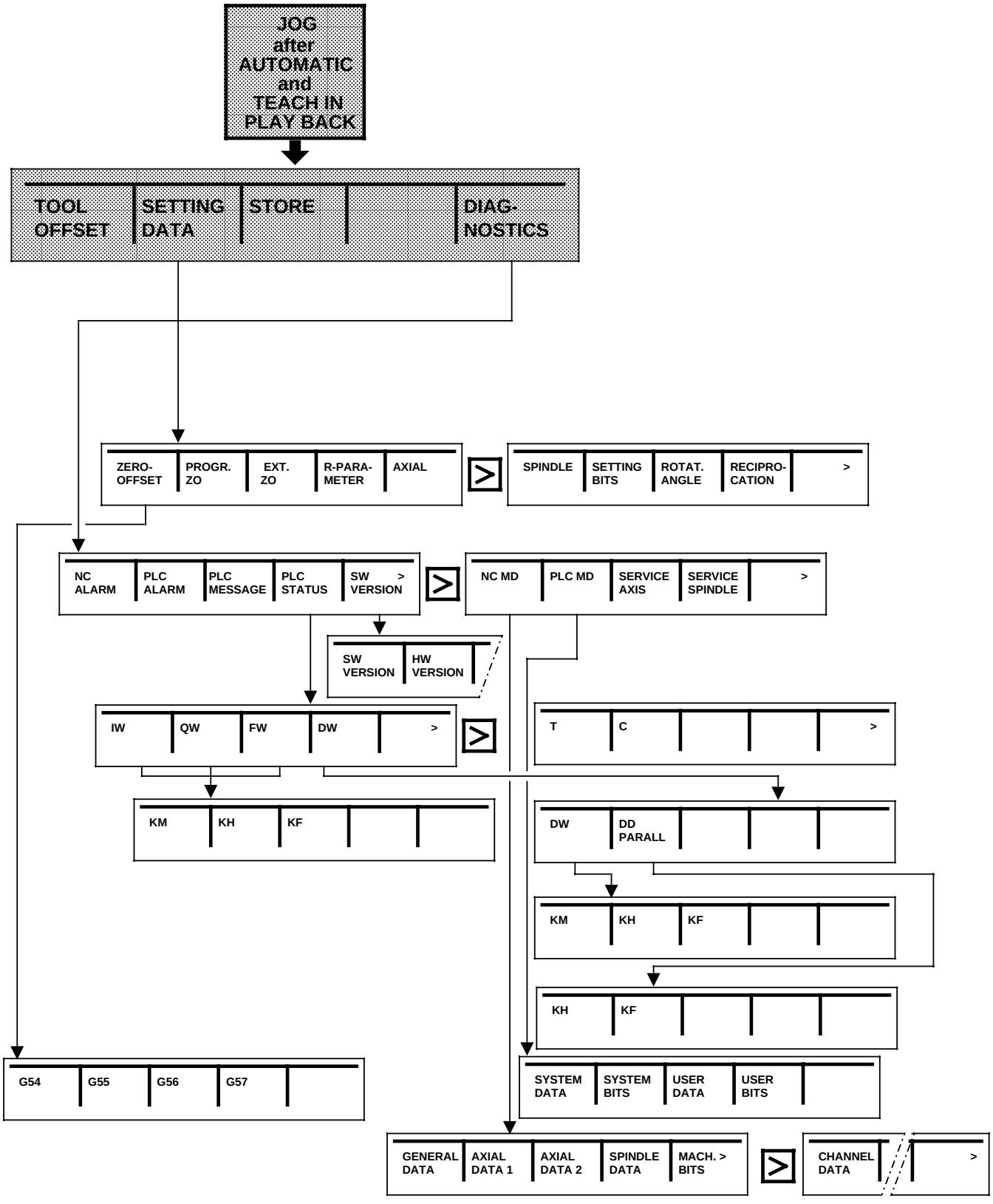


2.4.2 "JOG" mode

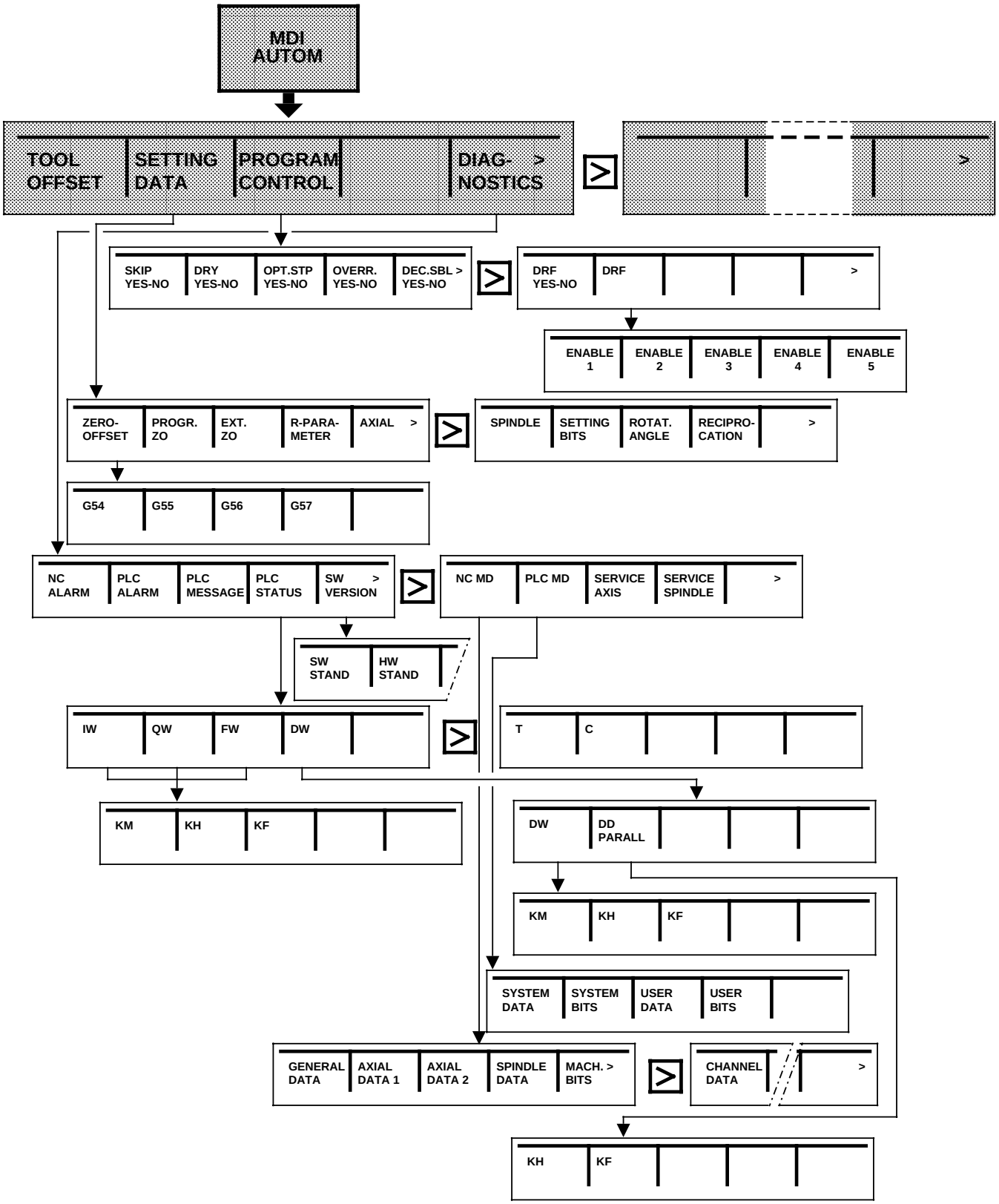




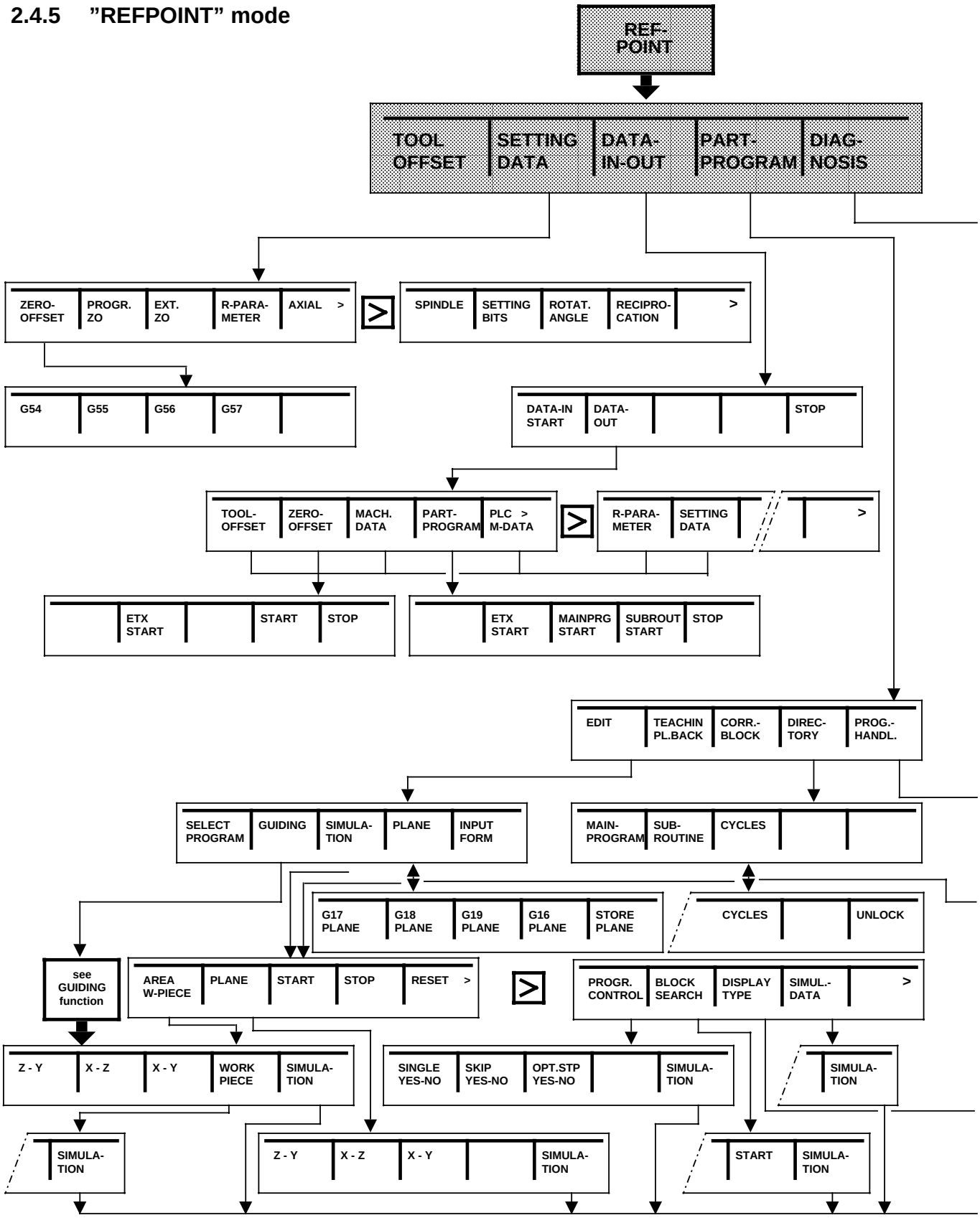
2.4.3 "JOG" mode (after "TEACH IN/PLAYBACK" in "AUTOMATIC" mode)

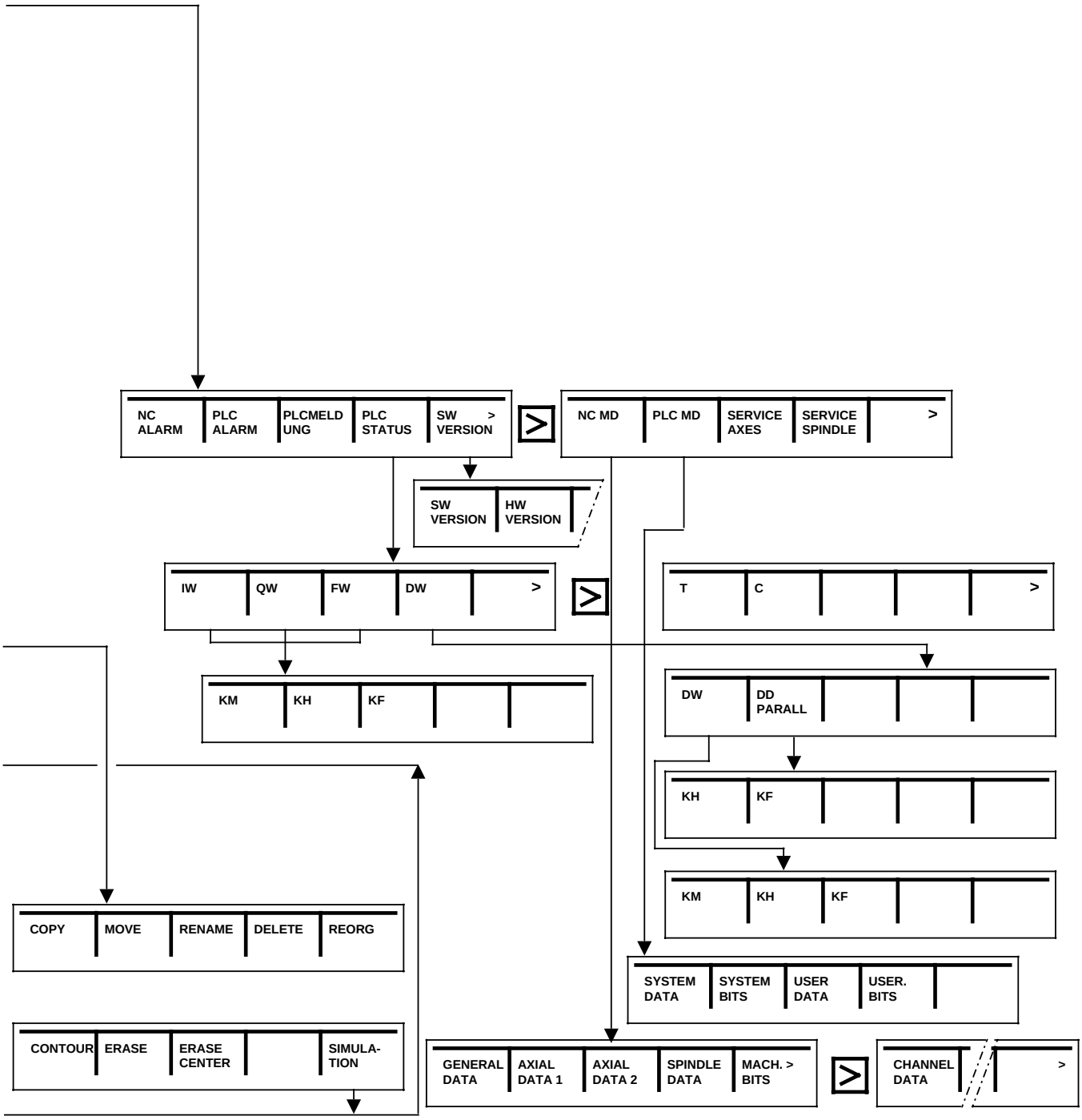


2.4.4 ”MDI AUTOMATIC” mode

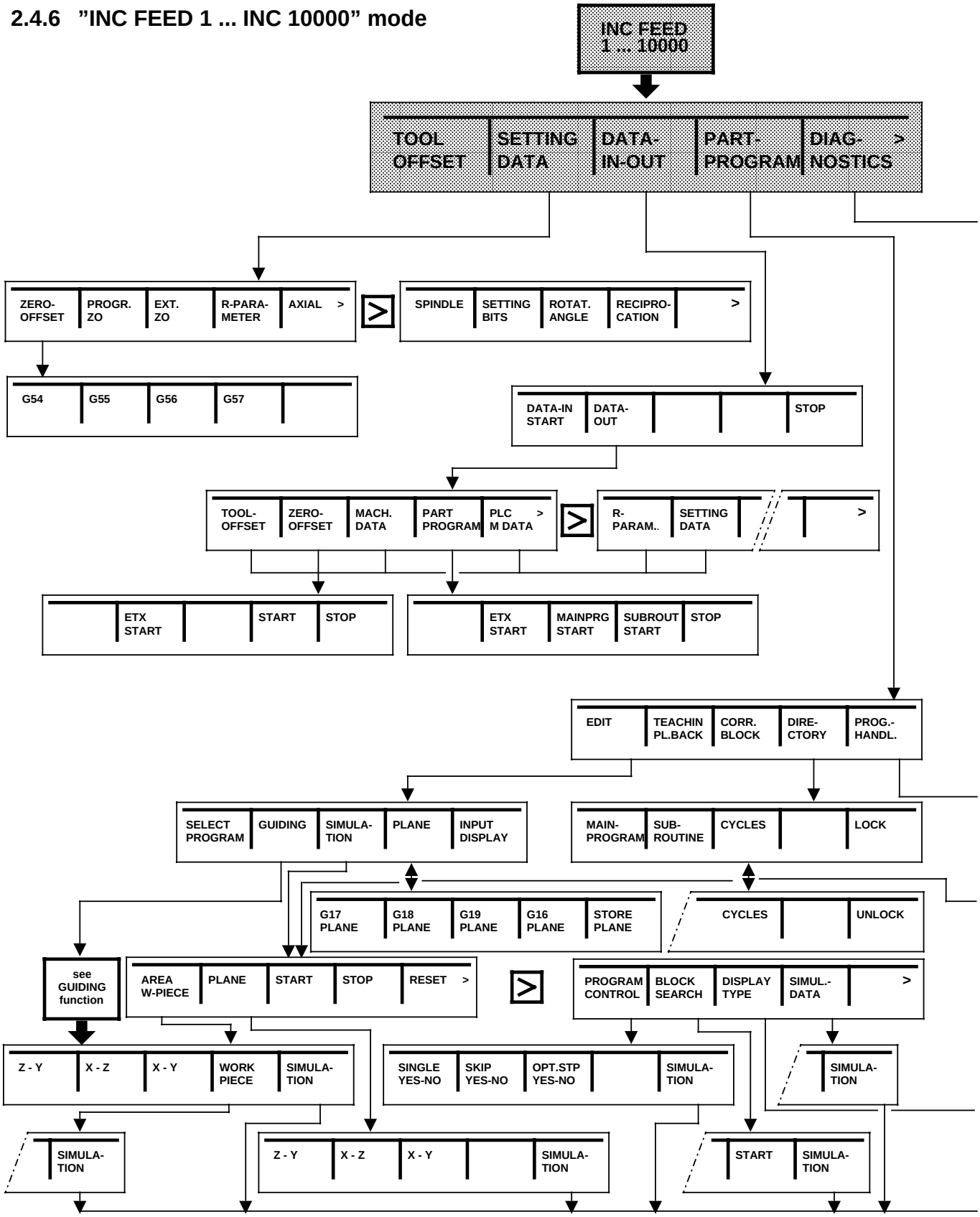


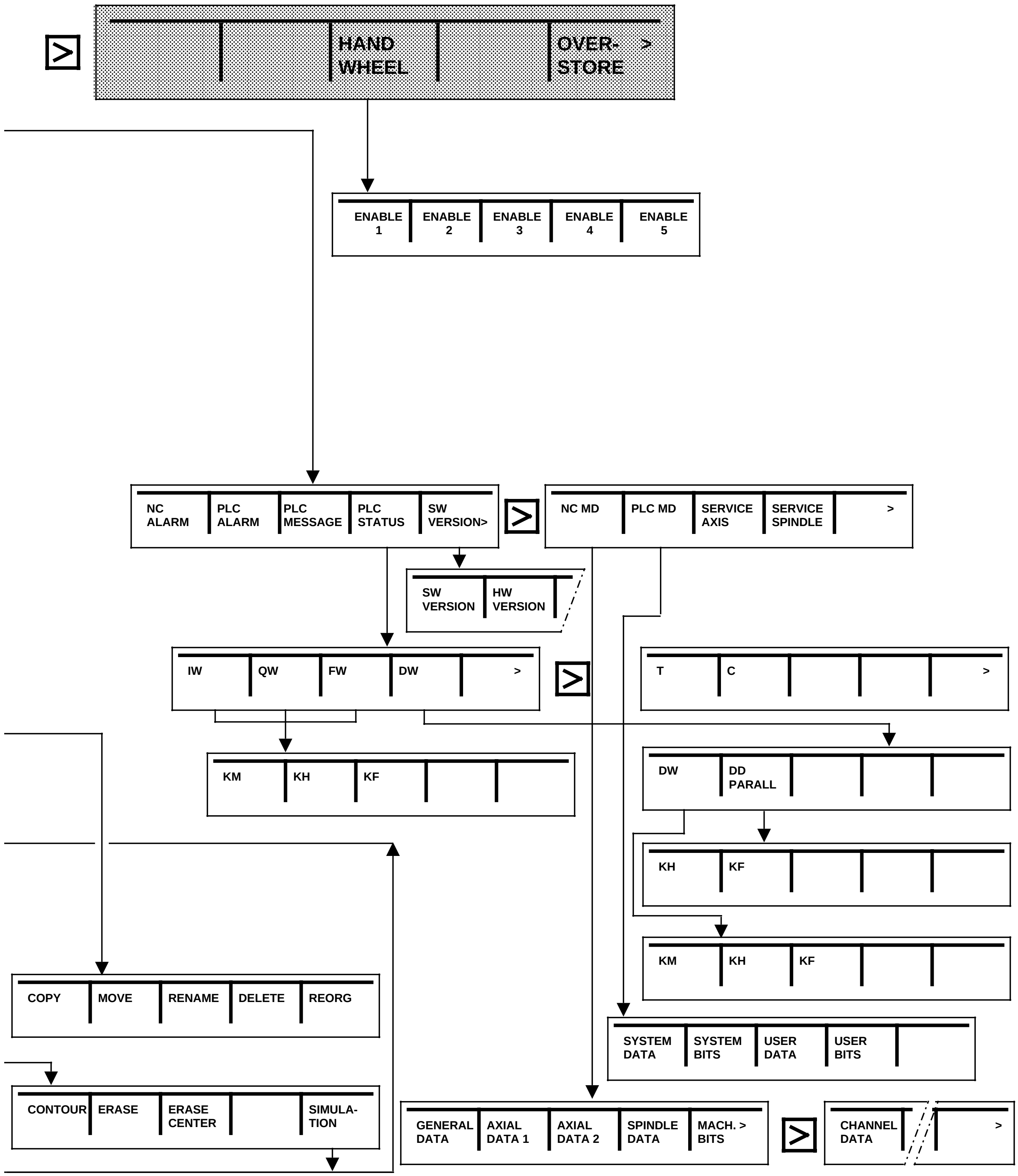
2.4.5 ”REFPOINT” mode



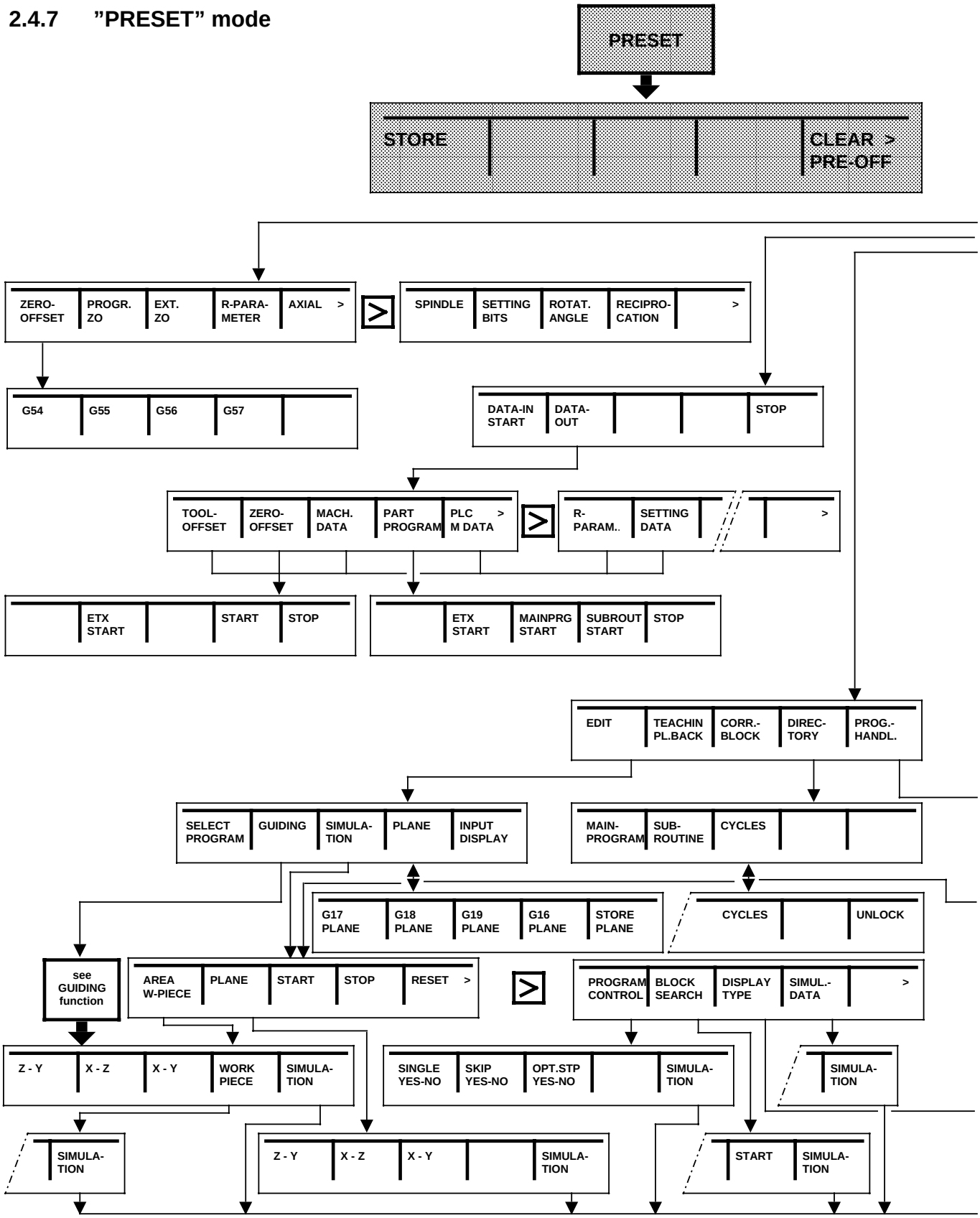


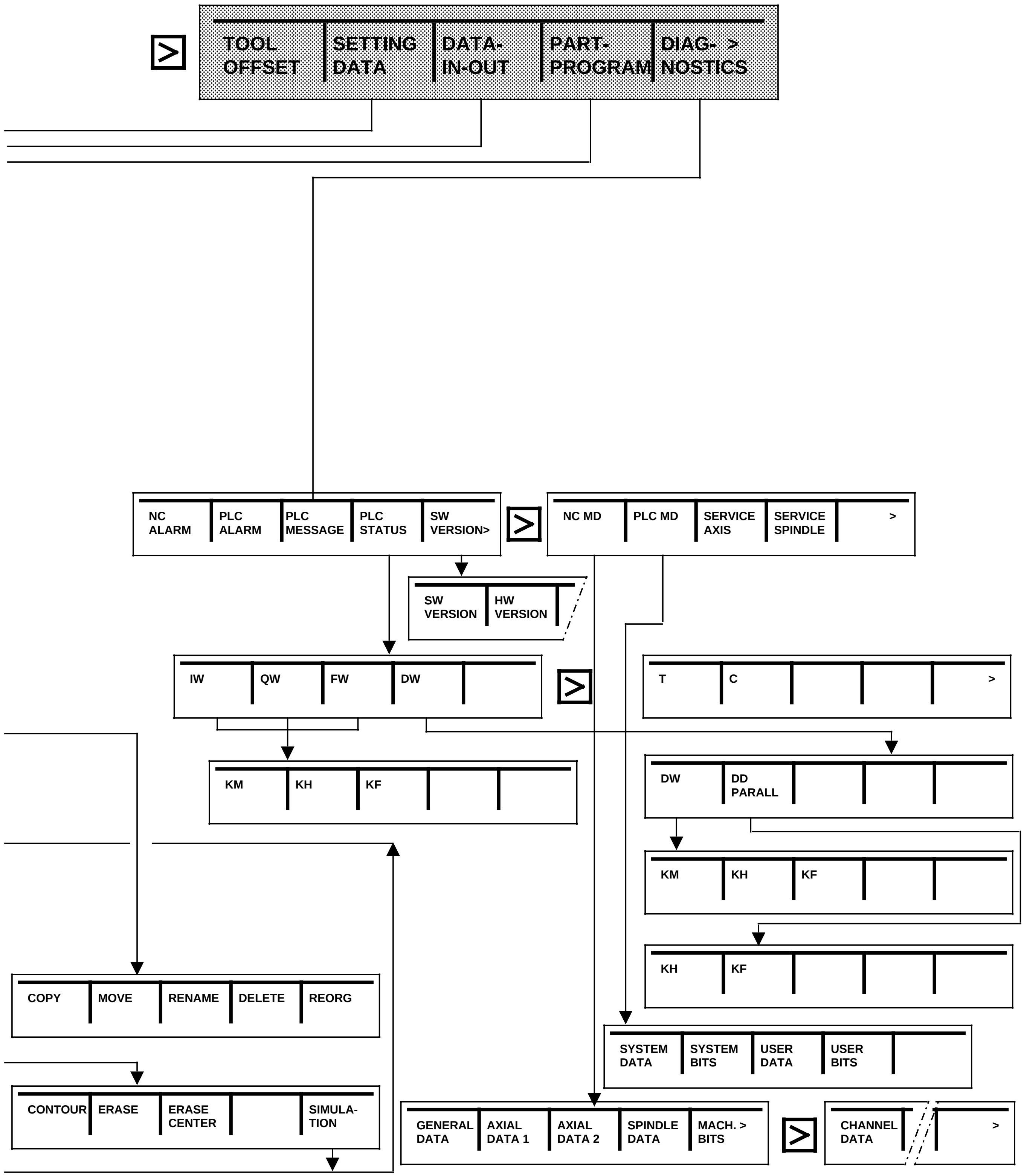
2.4.6 "INC FEED 1 ... INC 10000" mode



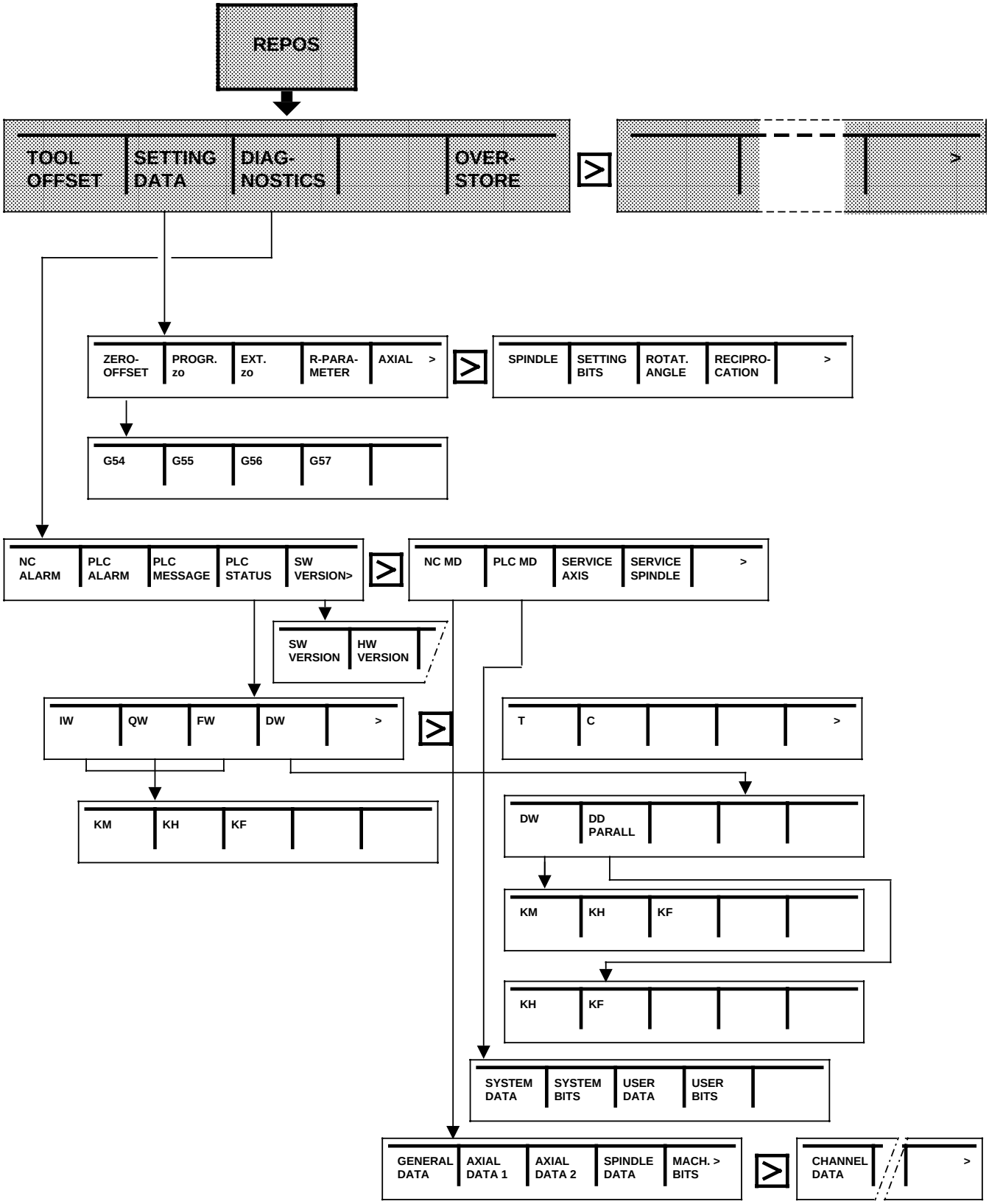

HAND WHEEL
OVER- STORE >
ENABLE 1
ENABLE 2
ENABLE 3
ENABLE 4
ENABLE 5
NC ALARM
PLC ALARM
PLC MESSAGE
PLC STATUS
SW VERSION>
NC MD
PLC MD
SERVICE AXIS
SERVICE SPINDLE
>
SW VERSION
HW VERSION
IW
QW
FW
DW
>
T
C
>
KM
KH
KF
DW
DD PARALL
KH
KF
KM
KH
KF
COPY
MOVE
RENAME
DELETE
REORG
SYSTEM DATA
SYSTEM BITS
USER DATA
USER BITS
CONTOUR
ERASE
ERASE CENTER
SIMULA- TION
GENERAL DATA
AXIAL DATA 1
AXIAL DATA 2
SPINDLE DATA
MACH. > BITS
CHANNEL DATA
>

2.4.7 "PRESET" mode

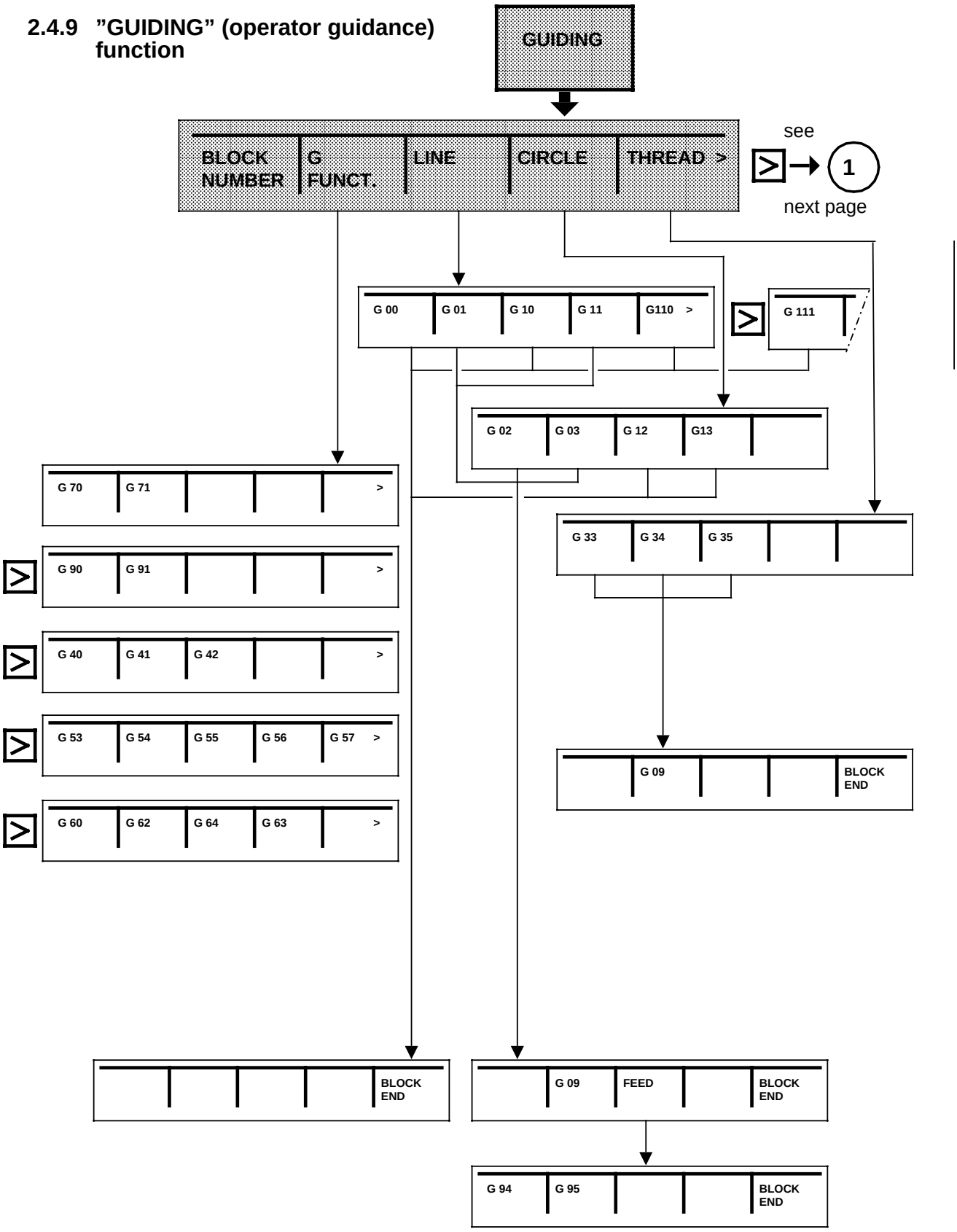



TOOL OFFSET
SETTING DATA
DATA-IN-OUT
PART-PROGRAM
DIAG-NOSTICS >
NC ALARM
PLC ALARM
PLC MESSAGE
PLC STATUS
SW VERSION>
NC MD
PLC MD
SERVICE AXIS
SERVICE SPINDLE
>
SW VERSION
HW VERSION
IW
QW
FW
DW
T
C
>
KM
KH
KF
DW
DD PARALL
KM
KH
KF
KH
KF
COPY
MOVE
RENAME
DELETE
REORG
SYSTEM DATA
SYSTEM BITS
USER DATA
USER BITS
CONTOUR
ERASE
ERASE CENTER
SIMULA-TION
GENERAL DATA
AXIAL DATA 1
AXIAL DATA 2
SPINDLE DATA
MACH. > BITS
CHANNEL DATA
>

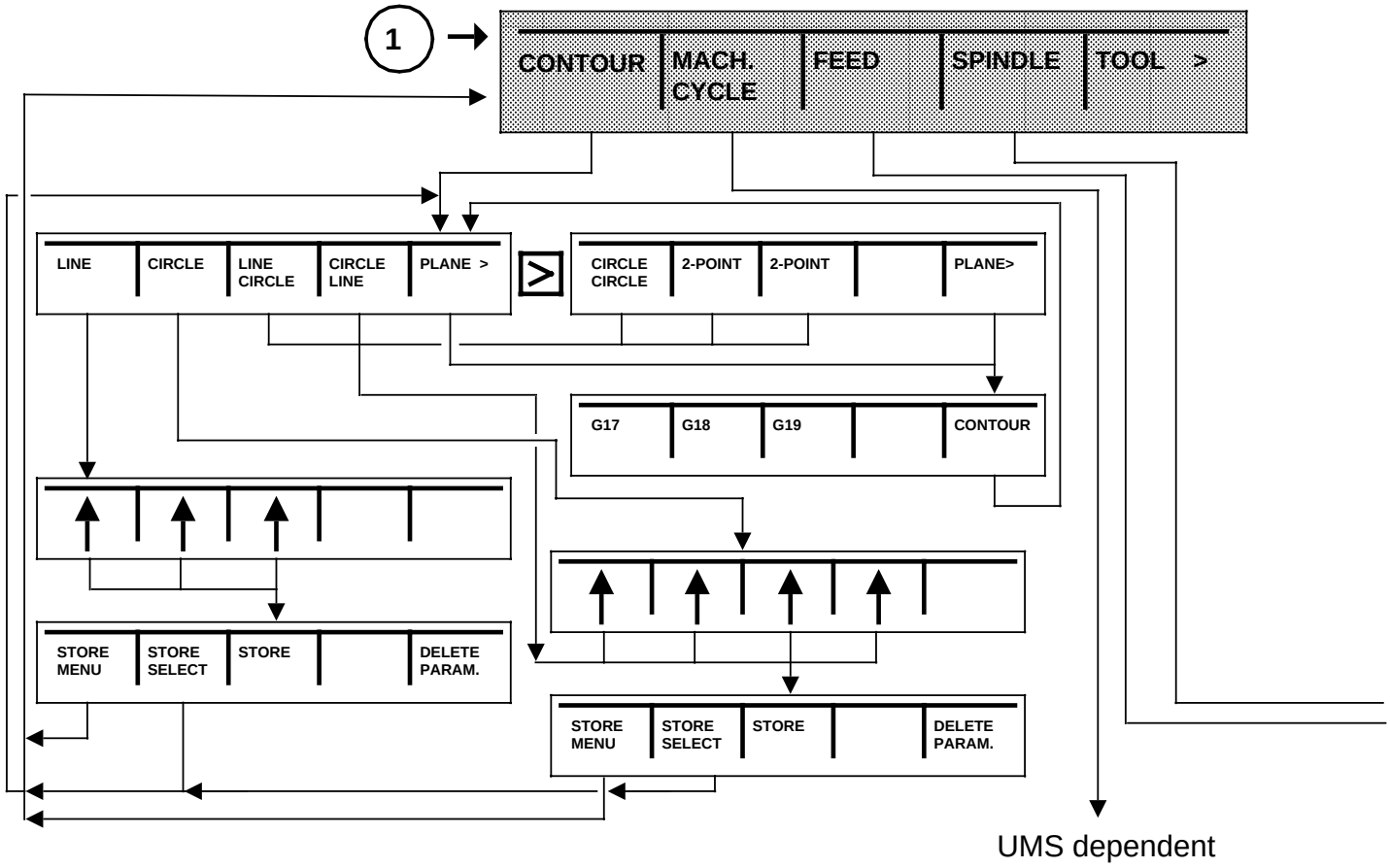
2.4.8 "REPOS" mode

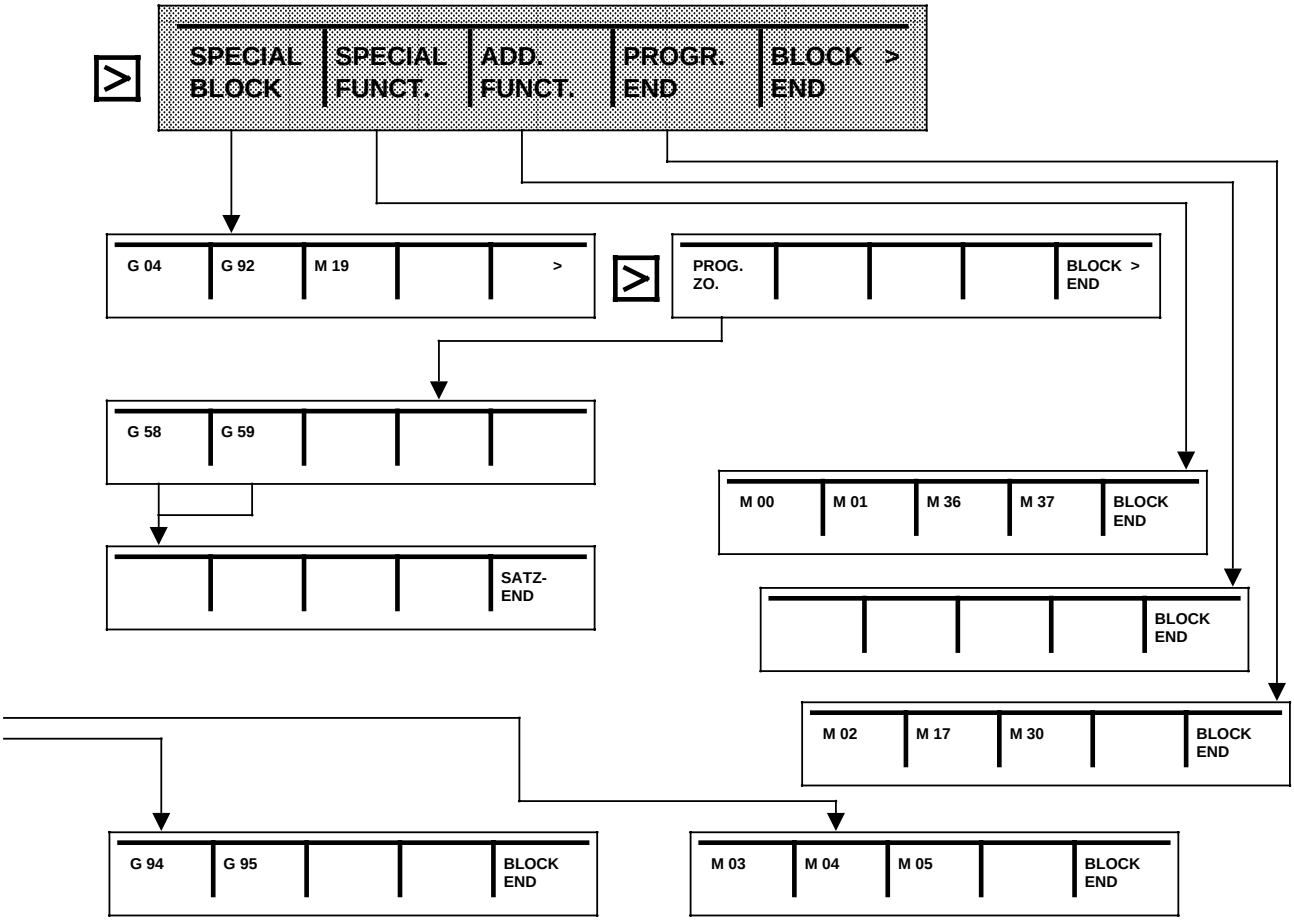


2.4.9 "GUIDING" (operator guidance) function



"GUIDIN" (operator guidance) function, continued





2.5 Glossary of softkey functions

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

ACTUAL BLOCK

(CURRENT BLOCK)

The current block is the block currently being processed. The display shows:

- The block **before** the current block, the current block and the block **after** the current block.

ACTUAL VALUES

(CURRENT VALUES)

The display shows the values for the current operating sequence.

ADD. FUNCTION

(ADDITIONAL FUNCTIONS)

The additional functions (fifth M group) fixed by the machine tool manufacturer are grouped together in this function. First press: softkey ”GUIDING”.

AREA W-PIECE

(WORKPIECE AREA)

Allows you to enter the simulation area. To input the workpiece dimension, press the ”AREA WORKPIECE” softkey.

BLOCK END

You select the character for ”End of Block (LF)” with this softkey.
First press: ”GUIDING” softkey

BLOCK NUMBER

The block numbers are automatically generated by the control in steps of five.

BLOCK SEARCH

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

CLEAR PRE-OFF

(CLEAR PRESET-OFFSET)

Clears the PRESET offset.

CIRCLE

Gives you the softkeys for selecting the G functions for circular interpolation.
First press: ”GUIDING” softkey (operator guidance).

CONTOUR

- During simulation: The programmed contour is displayed.
- During guiding: The ”contour” softkey function simplifies programming of workpiece contours with the transfer of values directly from the drawing and graphic displays.

You can select the following elements and combinations:

- Line
- Line circle
- Circle
- Circle line
- Circle circle
- 2-point definition
- 2-angle definition

COPY

Copies a part program and re-enters it into memory under another program number.

CORR. BLOCK

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

CYCLES

The following are displayed:

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

Cycles are protected subroutines which can be called for frequently used technologies or for machine-specific operations (stock removal cycles, drilling cycles, tool change cycles).
The values required are defined as parameter assignments before the cycles are called (cycle parameter assignment).

DATA-IN START

(START DATA INPUT)

Starts data input via the universal interface.

DATA IN-OUT

The data (part program, settable zero offsets, machine data, R parameters, tool offsets, PLC program, PLC operator messages, PLC machine data, user programs) are read in from an external device (e.g. punched tape reader) or output to an external device (e.g. printer). Data input and output is handled via interface 1 or 2. Interface 1 is variable, interface 2 is normally assigned to a particular device. The interface data can be defined separately using setting data.

DATA OUT

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

DEC-SBL YES-NO

(DECODING SINGLE BLOCK ON-OFF)

If ”YES”, the blocks are processed one by one. The function is active at the end of the block in which decoding takes place with the signal present (program control).

DELETE

Using ”DELETE”, you can clear one or more part programs from the program memory.

DIAGNOSTICS

All current alarms are displayed separately as NC alarms, PLC alarms and PLC messages. Other displays are for service purposes.

DIRECTORY

The following is displayed:

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

DISPLAY TYPE

(TYPE OF REPRESENTATION)

You can select either ”CONTOUR” or ”ERASE”.

DRF

The differential resolver function produces an additional incremental zero offset.

DRF YES-NO

If ”YES”, the differential resolver function is activated.

DRY RUN YES-NO

(DRY RUN REEDRATE YES-NO)

With ”YES”, the axes are traversed at the dry run feedrate, not the programmed feedrate. The dry run feedrates is set in setting data.

EDIT

EDIT gives you the softkeys ”SELECT PROGRAM”, ”GUIDING” and ”SIMULATION”.
Editing means:

- Input of a program into the memory or changing or altering a program that is already in the memory.

ERASE

The programmed machining process is displayed taking the tool data into account (e.g. cutter radius).

ETX START

Output of end of transmission character via the selected RS 232 C (V.24) interface.

FEED

Gives you the softkeys for selecting the G functions for the type of feed. First press: softkey ”GUIDING”.

G FUNCTIONS

Gives you the softkeys for selection the G functions from the groups G 7 to G 15 (see Programming Guide) .
First press: ”GUIDING” softkey.

GUIDING

Operator guidance speeds up and simplifies the input of part programs.
Apart from geometric functions (G function, contour definition) you can also input machining cycles and technological functions (feedrate, spindle speed) via softkeys.

HANDWHEEL

In the INC1 to INC10000 operating mode you activate an electronic handwheel via softkey.
The electronic handwheel enables operation which is equivalent to jog handwheel operation.

H-WHEEL **AX.

(HANDWHEEL ** AXIS)

With this function, you assign handwheels to axes.

LINE

Gives you the softkeys for selecting the G functions for linear interpolation.

MACH. DATA

(MACHINE DATA)

The machine data (MD) are used to interface the control to the machine. Certain functions are defined with machine data (e.g. ”Dry run feedrate interlocked with keyswitch”).
A distinction is made between NC MD and PLC MD. NC MD are divided up into general data, axial data and spindle data; the PLC MD are divided up into system data and user data.
Machine data are fixed on installation and cannot be changed by the user.
You select the output of machine data by pressing the ”MACH. DATA” softkey after ”DATA OUT”.

MAINPRG. START

(MAIN PROGRAM START)

Activates the output of part programs via the universal interface.

MOVE

Shifts the selected part program to the end of memory. Only the program at the end of memory can be edited while as another program is being executed.

NC ALARM

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

NC MD

(NC MACHINE DATA)

The NC machine data are displayed.

OPT. STOP YES-NO

(PROGRAMMED STOP ON-OFF)

If ”YES”, execution of the program is stopped at the point at which the ”M01” command is programmed.

OVERR. YES-NO

(FEED/RAPID OVERRIDE ON-OFF)

If ”YES”, the set override is valid for the rapid traverse override/feedrate override switch.

OVERSTORE

Allows you to change the values of T, D, S, H, M in the buffer memory.

PART PROGRAM

The ”PART PROGRAM” softkey gives you the following solftkeys:

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

This key is not used to select a program for execution.
After ”DATA OUT”, you can select programs output with the softkey ”PART PROGRAM”.

PLANE

Allows you to select the machining plane for simulation and blueprint programming.

PLAYBACK

While this function is active positions approached are stored to build up a program.

PLC ALARM

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

PLC MESSAGE

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

PLC MD

(PLC MACHINE DATA)

The PLC machine data are displayed. After ”DATA OUT”, you can select the output of PLC machine data with the ”PLC MD” softkey.

PLC STATUS

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

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

PROGRAM CONTROL

The key leads to the following functions:

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

PROGRAM END

Gives you the softkeys for selecting the functions of the ”M02” (end of program) group.

PROG. HANDL.

(PROGRAM HANDLING)

Allows you to copy, move to the end of the memory, rename and delete the program in the program memory. You can reorganise the program memory (after deleting programs).

R PARAMETER

Allows you to input the R parameters as setting data.

- After ”DATA OUT”, you can select the output of R parameters with the ”R PARAMETER” softkey.

RENAME

Allows you to change the program number. The program itself remains unchanged.

REORG

(REORGANIZE)

Allows you to reorganize the program memory. The space that has been made available by the deletion of programs can be used again for entering further programs.

RESET

Simulation is interrupted and returned to the reset state.

ROTAT. ANGLE

(ANGLE OF ROTATION)

Input of the setting data for coordinate rotation.

RESET

Simulation is interrupted and returned to the reset state.

SERVICE AXES

In this display, all the axis service data are displayed. With the ”page” key you can switch to the desired axis.

SETTING DATA

Using setting data, the operator (user) sets certain operating states. Setting data are used to set the following:

- Zero offsets
- Additive zero offsets
- R parameters
- Axial data
- Spindle data
- General data (setting data bits)

SIMULATION

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

SIMUL. DATA

Allows you to enter the data required for ”ERASE” representation.

SINGLE BLOCK YES-NO

After ”program start” only one block is executed. The next block is only executed after you have pressed ”program start” again.

SKIP YES-NO

If ”YES” is selected, the blocks marked with an oblique (/), are skipped during program execution (program control).

SPECIAL BLOCK

Gives you the softkeys for selecting G04, G92, M19, G58, G59

SPECIAL FUNCT.

The special functions M00, M01 (first M group) and M 36, M37 (fourth M group) are grouped together in this function.

SPINDLE

Gives you the softkeys for selecting the M function for the spindle motion. First press: ”GUIDING” softkey.

START

Activates the selected softkey function.

STOP

Stops the activated softkey function.

STORE

In ”JOG” mode you store the position traversed to.

STORE AXIS

Stores the axis-specific PRESET values entered.

STORE DRILL PATTERN

Stores the values entered and return to the selection menu.

STORE MENU

Store the values entered and return to the basic menu.

STORE SELECT

Stores the values entered and return to the selection menu.

SUBROUT.

Displays:
- The stored subroutine numbers
- The number of characters used
- The free memory space

TEACH IN

In ”TEACH IN” mode you generate a part program block by block. By doing this, you can test the program immediately.

THREAD

Gives you softkeys for selecting the G functions for threading.

TOOL

Allows you to input the tool number T... and the tool offset number D... .

TOOL OFFSET

The tool offset takes into account the tool dimensions and wear. The tool offset is stored under a tool offset number, D1 to D99, in the tool offset memory.
By pressing the ”TOOL OFFSET” softkey after ”DATA OUT”, you can select the output of tool offsets via the universal interface.

WORK CYCLE

(MACHINING CYCLE)

Gives you softkeys for selecting cycles for frequently occuring machining sequences.
First press: ”GUIDING” softkey.

WORKPIECE

Allows you to input the workpiece dimensions in the input screen form.

ZERO OFFSET

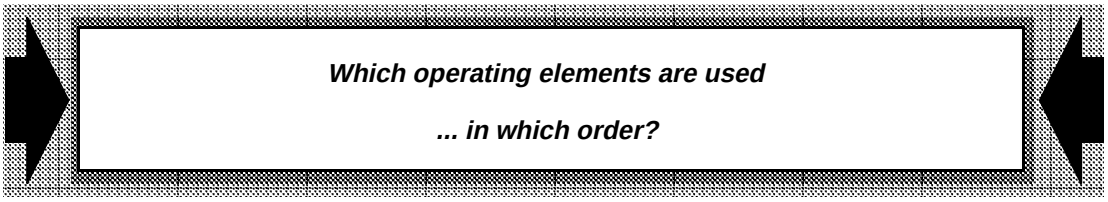
The settable zero offsets are input as setting data. Programmable and external zero offsets are displayed on the CRT (see Section 3.1.5).

3 Operating Sequences

3.0 Preliminary remarks

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

This section shows how to use the operating elements in frequently occurring operating sequences.
It explains:



The section was 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 machining.
- In the second part (operating) the operating methods are explained that may be required during **machining**.
- Where **dual function keys** (address/numberical keys) are shown in the following subsections, the part of the illustrations **shaded grey** shows the function required.

3.1 Preparation

3.1.1 Switching on

When you switch the control and machine on:

Follow the instructions in the machine tool manufacturer's documentation!

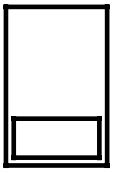
3.1.2 Traverse to reference point

After switching on you traverse the individual axes to their reference points. This synchronizes the control with the machine.

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

Sequence of operation

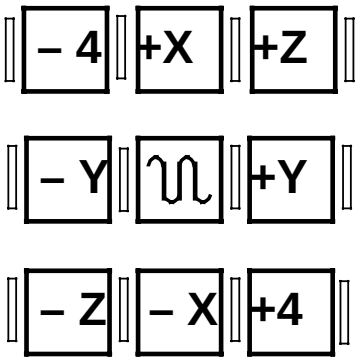
REF-POINT



Select the ”REFPOINT” operating mode with this softkey

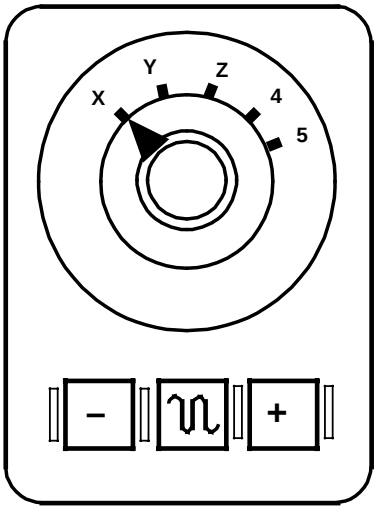
or

Rotate the operating mode selector switch on the external machine control panel to this symbol (REFPOINT).



Press the direction keys on the integrated machine control panel

or



Select the axis you wish to traverse with the axis selector switch on the external machine control panel and then press the direction key. You can now traverse the individual axes one after the other to their reference points.

Note:

The selected traverse direction is checked by the control before starting:
If you have pressed the wrong direction key (say +X instead of –X),
no traverse motion will result as the operation is rejected.)

With the ”feed hold” key you can stop the selected axis before the reference point is reached.

aus 0HQ01, S. 3-4
Änderungen s. Anlagen

After reaching the reference point, the position values with reference to the machine zero point are displayed as ”ACTUAL POSITION” on the CRT. The control is now synchronized with the machine.

3.1.3 Tools

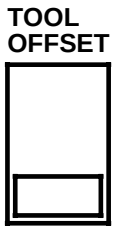
The machine tool manufacturer's documentation should be consulted about tooling.

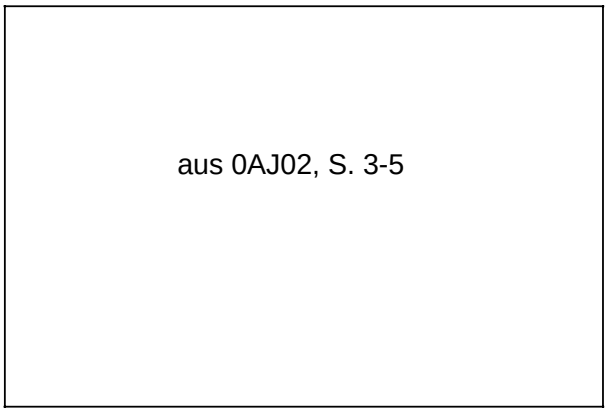
3.1.4 Tool offsets

3.1.4.1 Input of tool offsets

The input screen form will help you to input ”tool offsets”.

You can call up this screen form with this softkey in **all** operating modes.





This input screen form appears.

Explanation of the input screen form

D 5

Name of the tool offset memory (”D5” in this example).
The SINUMERIK 810G has 99 offset memories.
You can therefore address the tool offset numbers D1 ... D99, and store offset data under each one.

0 Tool number

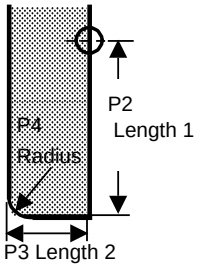
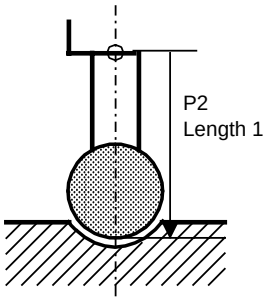
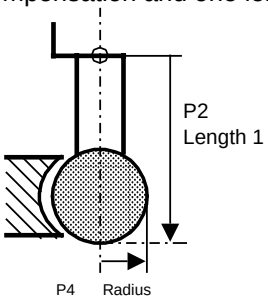
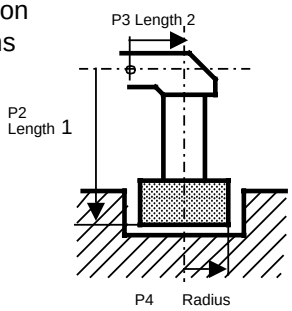
The input field marked ”0” is intended for the input of a max. eight-digit ”tool number”.
Entries are only required when using flexible tool management!

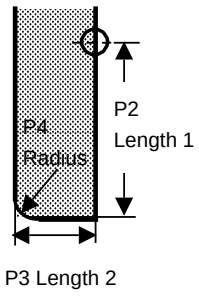
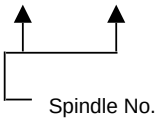
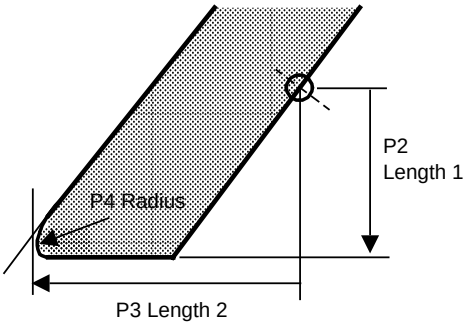
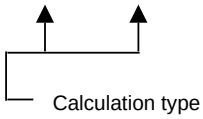
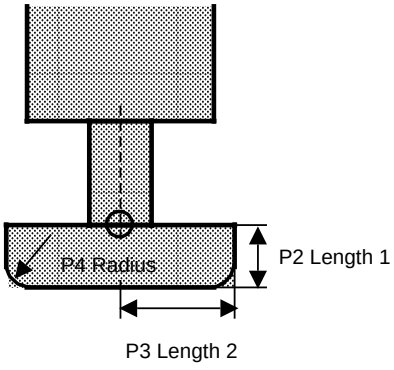
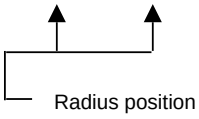
Whether tool numbers are used or not is set in MD!

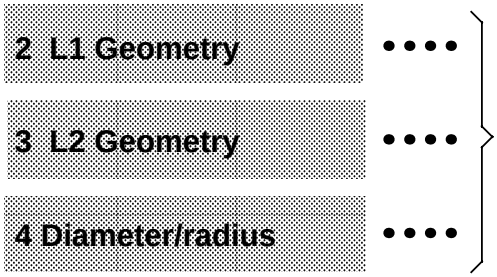
1 Tool type

The input field marked ”1” is reserved for the input of the ”tool type P1”.

Explanation of tool type:

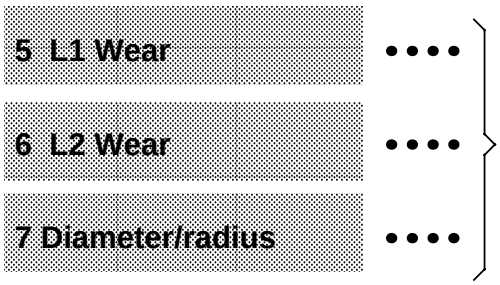
Tool	Tool type P1=. . .
Grinding wheel with radius compensation and two length compensations in the machining plane 	1 ... 9
Tools with length compensation only 	10
Tools with radius compensation and one length compensation 	20
Tools with radius compensation and two length compensations 	30

Tool	Tool type P1=. . .
Grinding wheel with radius compensation and two length compensations and **monitoring parameters** 	161 ... 169 261 ... 269 
	151 ... 159 251 ... 259 
	171 ... 179 271 ... 279 
Spindle No. 9 is a special identifier for a dresser with radius compensation and two length compensations in the machining plane. As for types 1 to 9, however, if the monitoring functions were active for the previous tool they will remain active.	901 ... 909



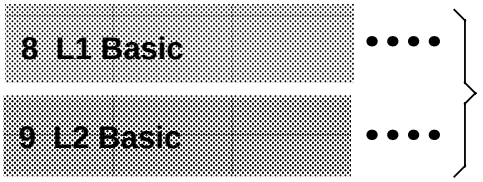
The input fields marked **"2"** and **"3"** are for the geometry values of the tool.
The input field marked "**4**" is for the value for the cutter radius.

Input of tool geometry values can be prevented by keyswitch.



In the input fields marked **"5"** to **"7"**, you can enter the wear data of the tools into the input form if you wish (not obligatory)

Input of the wear data can be prevented by keyswitch.



The input fields marked "**8**" and "**9**" are provided for special applications.
The "basic dimension" permits an additional tool length compensation.

Tool parameters P10 to P15 are not shown in the standard display. A more detailed text display can be implemented by configuring several tables.

10 Following D No. ....

The data of a grinding wheel can be stored in several D fields. Parameter 10 points to the next D field for the same tool.

11 Min. diameter ....

12 Min. width ....

13 Current width ....

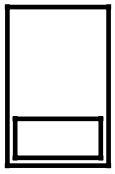
14 Max. speed ....

15 Max. surface speed ....

In the input fields marked "**11**" to "**15**", you can enter monitoring parameters for the tool in the input screen form. The monitoring parameters are taken into account if the tool type is between 151 and 909.

Sequence of operation

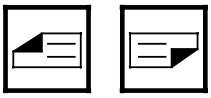
TOOL OFFSET



Press the "tool offset" softkey.
(This is possible in all operating modes.)

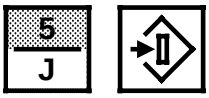
Bild aus 0AJ02, S. 3-9

This input screen form appears.
The current input field (the field into which your input will be transferred) is displayed in inverse video.

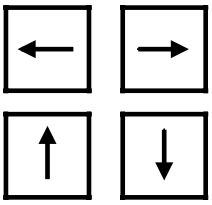


Select the desired tool offset number, in the range D1, ... D99, by pressing either the ”page forward” or ”page backwards” key

or

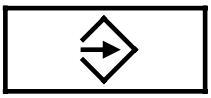


Call the desired tool offset number D ... (”5”, in this example).
The letter ”D” need not be entered!
Now press the search key.



Now press any of the cursor keys to position the cursor in the desired input field (Field 0 to 15).

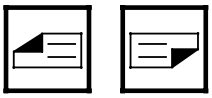
Now enter the offset value you require (”153” in this example) with the numeric keys.
You can see the entered value on the **input line** in the display.



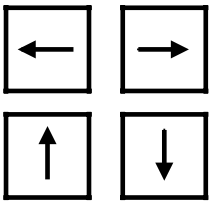
Now press the yellow input key to store the offset value in the offset memory.
The value you have input is now displayed in the selected **input field**.

- The inverse video marking of the input field now jumps to the next position (next offset value P ..) of the selected offset number D ...
- When you have stored **all** the offset values of the selected offset number D..., the inverse video marking of the input field jumps to the **first** offset value (P ..) of the **following** offset number (D ..).

3.1.4.2 Deleting/modifying an individual offset value



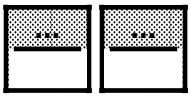
Page to the desired tool offset number under which you intend to delete/modify an offset value.



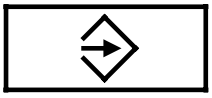
With these keys, position the cursor on the input field to be deleted/modified.

Using the numeric keys, enter the value ”zero” (delete)

or



using the numeric keys, enter a new value (modification).



With this key, transfer the deletion or modification into the offset memory.

3.1.4.3 Deleting all offset values under an offset number D...

a) You would like, for example, to delete all values stored under tool offset number ”D5”:

Sequence of operation

Using the key, enter ”5”.

Press the ”delete word/block” key.

When you now select the tool offset number ”D5”, using either the ”page” keys,   or by entering the number ”5” and pressing the  search key, you can check that all the values under this offset number have been set to ”zero”!

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

Sequence of operation

Using these keys, enter ”4 = 9”.

Press the ”delete word/block” key.

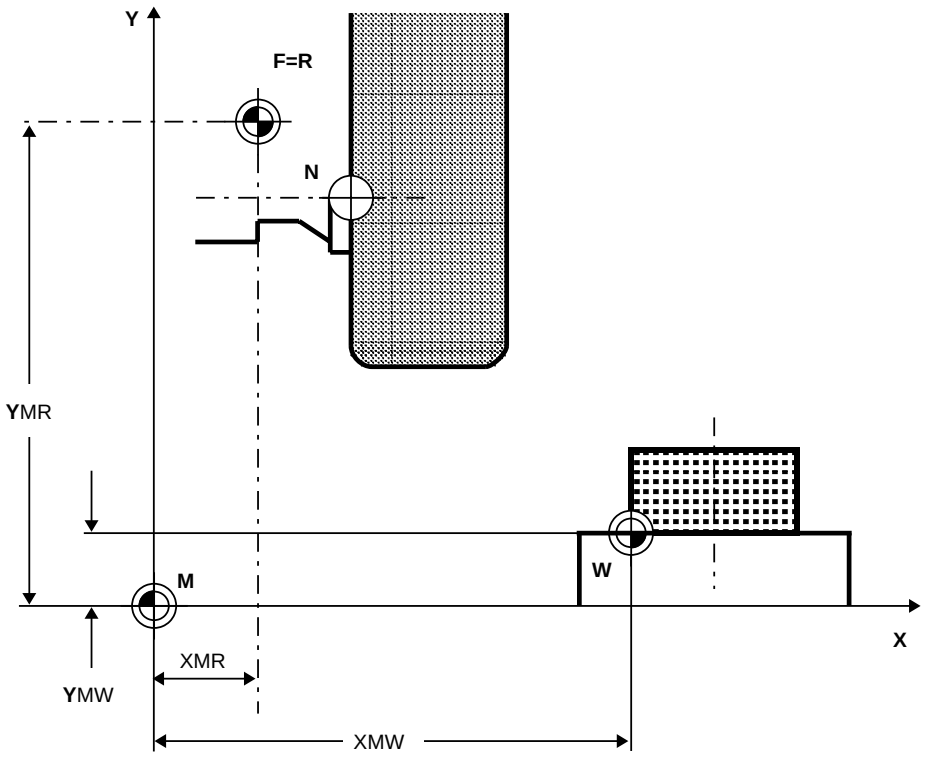
When you now call the tool offset numbers ”D4” to ”D9”, as described in a) above, you can check that all offset values have been set to ”zero”!

3.1.5 Zero offset

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

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

Machine zero point **"M"** and workpiece zero point **"W"** are not identical. The dimensions between the machine zero point **"M"** and the workpiece zero point **"W"** can vary, depending upon the type and fixing of the workpiece. The **zero offset** will be taken into account during program execution.



M	Machine zero point
W	Workpiece zero point
R	Machine reference point
F	Slide reference point
XMR, **Y**MR, etc.	Reference point coordinates
XMW, **Y**MW, etc.	Zero offsets
N	Tool holder point

3.1.5.1 Settable zero offset

For each axis you can select four variable zero offsets using ”G54” to ”G57”.

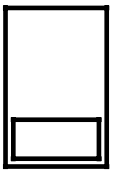
For ”G54” to ”G57” each have two settings :

- a ”coarse offset” and
- a fine offset/correction of the zero points (**ADD. ZO**)

You can set the values for the settable zero offset in the ”SETTING DATA”.
The entered zero point offsets are activated in the part program called.

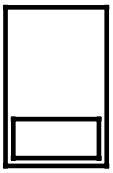
Sequence of operation

SETTING DATA



In any operating mode, press the ”SETTING DATA” softkey

ZERO OFFSET

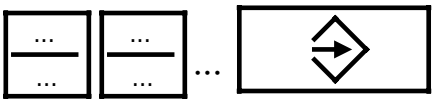


... and then the ”ZERO OFFSET” softkey.

aus 0HQ01, S. 3-14
Änderungen s. Anlage

This input screen form appears for the input of values for the ”coarse setting G54 to G57”.

The input field ”G54” for the **X axis** is marked in inverse video.

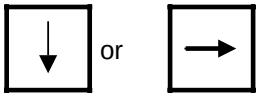


Now enter the value for the coarse offset ”G54” for the **X axis** into the input line, using the numeric keys; then transfer the value into the input field with the input key.

The inverse marking now jumps to the next input field (coarse offset for the Y axis).

Now enter the value for the coarse offset for the **Y axis** with the numeric keypad. Then transfer the value into the input field with the enter key.

The inverse marking now jumps to the next input field (coarse offset for 3rd axis).



Press this cursor key until the inverse marking jumps into the right-hand input field (**X**).

Now enter the value for the fine offset ”G54” for the **X axis** into the input line, using the numeric keys; then transfer the value into the input field with the input key.

The inverse marking now jumps to the next input field (fine offset for Y axis).

Now continue as described above until you have entered all values for the coarse and fine offsets for G54 to G57.

3.1.5.2 Programmable zero offset - external zero offset

a) **Programmable zero offsets**

The values for the G58 and G59 offsets can be written in the program.

You will find information on the programming of these zero offsets in the User Documentation for the SINUMERIK 810G (Part 2: ”Programming”).

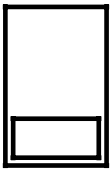
The programmed zero offset for the current part program % ... can be displayed.

Sequence of operation

Prerequisite:

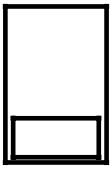
You have selected the ”AUTOMATIC” operating mode, either by softkey or by rotating the operating mode selector switch to the symbol  and you have called a program % ...

SETTING DATA



Now press the ”SETTING DATA” softkey

PROG ZO



... and then the ”PROGRAMMABLE ZERO OFFSET- EXTERNAL ZERO OFFSET” softkey.

aus 0HQ01, S. 3-16
Änderung s. Anlage

This display appears.
Now you can read the values for ”G58” and ”G59” for the X, Y, Z axes (and fourth and fifth axes if present).

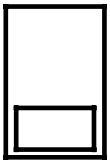
b) **External zero offset**

The ”External zero offset” values for the X, Y, Z axes (and fourth and fifth axes, if present) are transferred from the PLC.

These values can also be displayed on the screen.

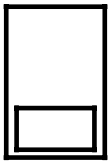
Sequence of operation

SETTING DATA



Press the softkey "SETTING DATA" in any mode ...

EXT.-ZO



... and then the softkey "EXT. ZO".

aus 0HQ01, S. 3-17
Änderung s. Anlage

This display appears.
You can read the external zero offset for the X, Y and Z axes (and fourth and fifth axes, if present).

The ”external zero offset” is set in machine data (MD).

3.1.6 Setting data :”R PARAMETERS”, ”SPINDLE”, ”AXIAL”, ”ANGLE OF ROTATION”, ”RECIPROCATION”

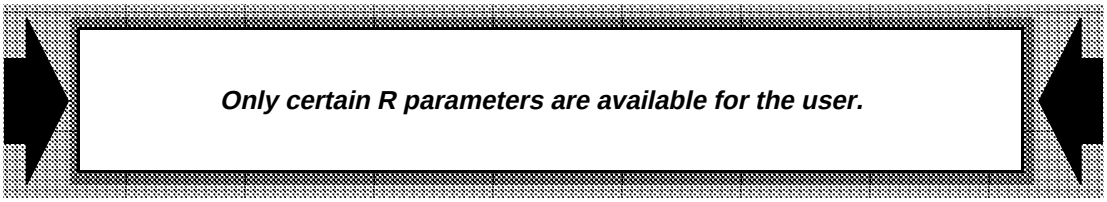
Using the **setting data**, you determine certain operating states.

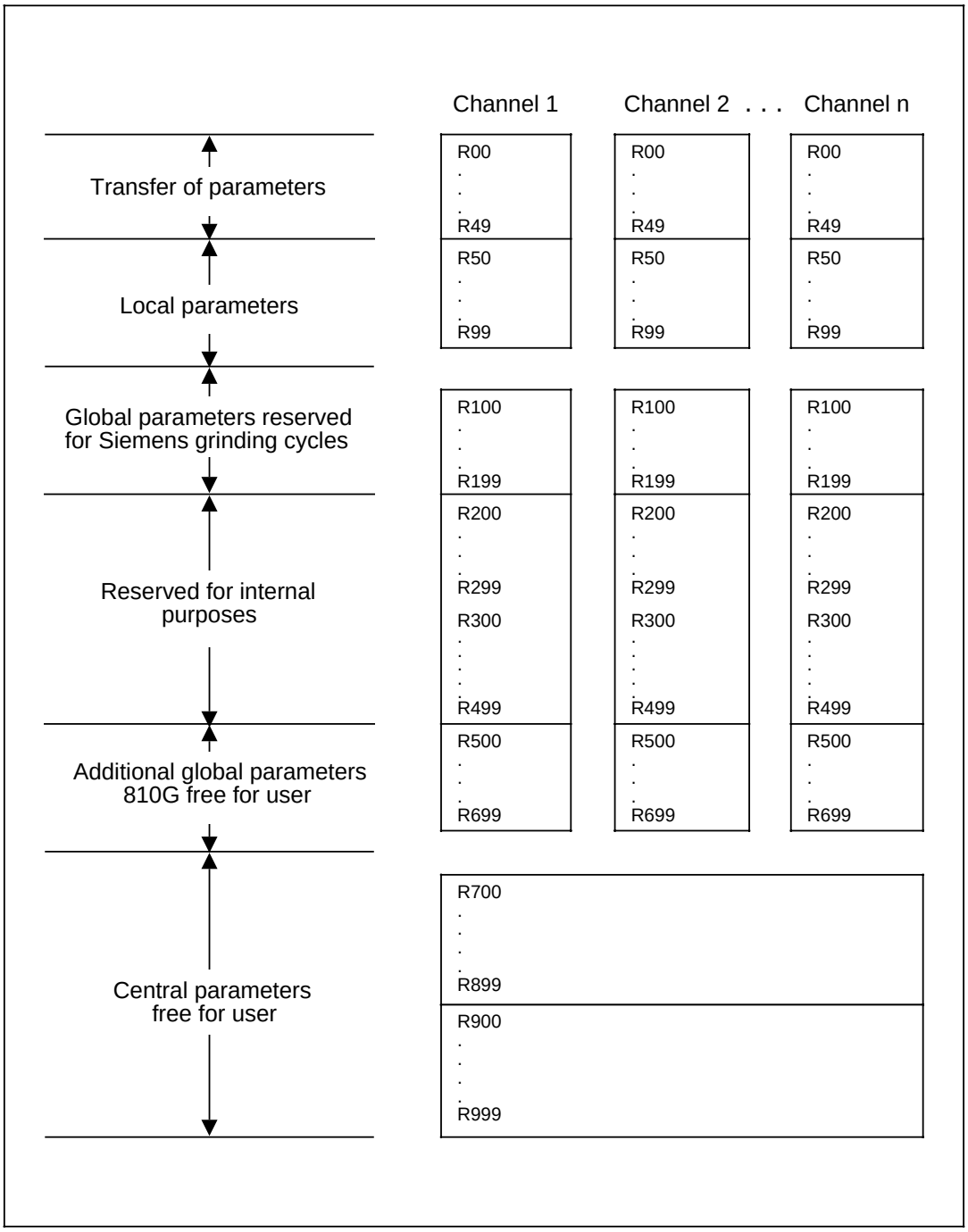
You can **input**, **modify** or **delete** setting data using screen forms.

Setting data ”R PARAMETERS”

The SINUMERIK 810G control recognises parameters R0 to R699 and R700 to R999.

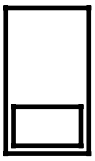
- Parameters R0 to R699 are channel-specific, i.e. they exist for every channel, ”CH1”, ”CH2” and ”CH3” separately.
- Parameters R700 to R999 are common to all channels (central parameters).




Channel 1
Channel 2 . . .
Channel n
Transfer of parameters
Local parameters
Global parameters reserved for Siemens grinding cycles
Reserved for internal purposes
Additional global parameters 810G free for user
Central parameters free for user
R00 . . . R49
R50 . . . R99
R100 . . . R199
R200 . . . R299
R300 . . . R499
R500 . . . R699
R700 . . . R899
R900 . . . R999

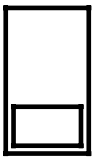
Sequence of operation for ”R PARAMETER” setting data

SETTING DATA



Press the ”SETTING DATA” softkey in any of the operating modes.

R-PARA-METER

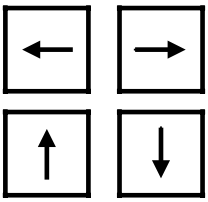


Press the ”R PARAMETER” softkey.

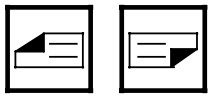
aus 0AJ02, S. 3-19
Änderung s. Anlage

This input screen form appears with the input field marked in inverse video.

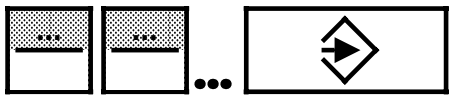
Using the ”channel switchover” key, select either channel ”CH1” or ”CH2” (marked in inverse video in the upper right hand corner of the display)



Now press any of the cursor keys to position the cursor on the desired input field of the displayed R parameters.



or



You can call up further screen forms with input fields for other R parameters with the page up/down keys

or

You can call up an input screen form or a particular R parameter using the alphanumeric keys and the search key ("63" in example, "R" need not be entered).

Enter the value for the selected R parameter with the numeric keys.
Then transfer the value into the input field with the input key.

"SPINDLE" setting data

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

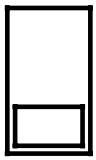
Designation	SD number	Standard setting data (as delivered)	Max. input value	Input units
Prog. spindle speed limitation	401*[1)]	0	16000	rev **/** min
Oriented spindle stop	402*[1)]	0	359.9	1 **/** 10 degr.
Spindle speed limitation (absolute)	403*[1)]	0	16000	rev **/** min

The maximum input value is set in MD.
The "input unit" is set in MD: you can also use 0.1 rev/min instead of rev/min!

*1) *= 0 for spindle 1*
1 for spindle 2

Sequence of operation for ”SPINDLE” setting data

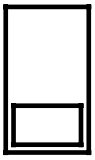
SETTING DATA



Press the ”SETTING DATA” softkey in any operating mode.

Extend the softkey menu displayed using the key on the right below the display.

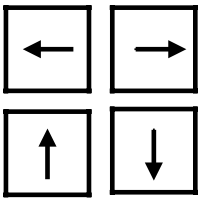
SPINDLE



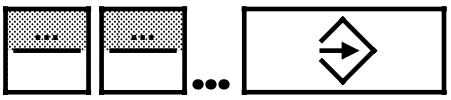
Press the ”SPINDLE” softkey.

aus 0AJ02, S. 3-21
Änderungen s. Anlage

This input screen form appears with the input field marked in inverse video.



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

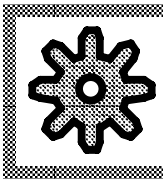


Enter the value for the selected input field with the numeric keys.
Then transfer the value into the input field with the input key.

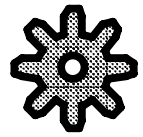
"AXIAL" setting data

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

Designation	Standard setting data (delivery state)	Maximum input value	Input unit
Smoothing constant for thread	0	5	–
Dry run feedrate	0	49000	1000 units/min (IS) 1)
Min. working area limitation	0	+/-99999999	mm/Inch 2)
Max. working area limitation	0	+/-99999999	mm/Inch 2)
Axis-handwheel assignment	0	0 ... 3	–

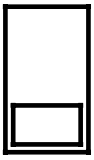


The "max. input value" is set in MD



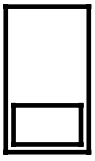
Sequence of operation for "AXIAL" setting data

SETTING DATA



Press the "SETTING DATA" softkey in any operating mode

AXIAL



Press the "AXIAL" softkey.

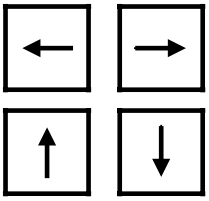
---

1) *You can obtain the maximum input value from the machine tool manufacturer (standard value 49000)*

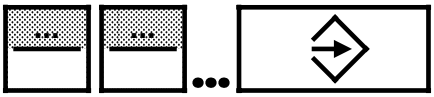
2) *Floating-point position*

aus 0HQ01, S. 3-24
Änderungen s. Anlage

This input screen form appears with the input field marked in inverse video.



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

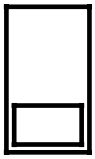


Enter the value for the selected input field with the numeric keys.
Then transfer the value into the input field with the input key.

"ANGLE OF ROTATION" setting data

Sequence of operation for "ANGLE OF ROTATION" setting data

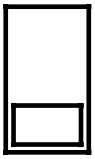
SETTING DATA



Press the "SETTING DATA" softkey in any operating mode.

Extend the softkey menu displayed using the key on the right below the display.

ANGLE OF ROTATION

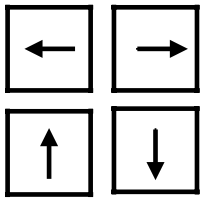


Press the "ANGLE OF ROTATION" softkey .

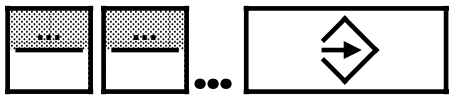
The "SETTING DATA COORDINATE ROTATION" input screen form appears.

Input the desired angles of rotation A... for G54 to G57 into the input screen form (the angles of rotation A... for G58 and G59 are defaulted by the program).

Do this as follows:



Position the input field on the desired position using the cursor keys (G54 to G59).

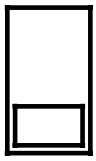


Enter a value for the selected angle of rotation "A" with the numeric keys.
Then transfer the value into the input field with the input key.

”RECIPROCATION” setting data

Sequence of operation for ”RECIPROCATION” setting data

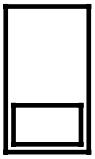
SETTING DATA



Press the ”SETTING DATA”softkey in any operating mode.

Extend the softkey menu displayed using the key on the right below the display.

RECIPRO-CATION



Press the ”RECIPROCATION” softkey.

aus 0HQ01, S. 3-26
Änderungen s. Anlage

The ”SETTING DATA RECIPRO-CATION” input screen form appears.

3.1.7 Program input

You can input a program by

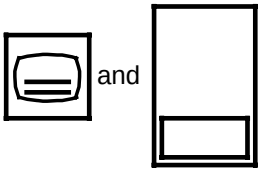
- using the keyboard
- using operator guidance.

While entering a program using one of the above methods, it is possible to change over to the other method; you can **alternate** between working with operator guidance and the keyboard (see also SINUMERIK 810G "User Documentation", Programming).

3.1.7.1 Program input with the keyboard

You enter an opened program % ... with the individual blocks N ... LF to N ...LF character by character only with the keyboard.

Sequence of operation

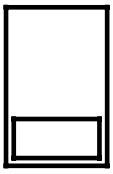


Call any of the following operating modes:

- PRESET (Actual value setting)
- JOG (Feed/Jog)
- AUTOMATIC (Automatic mode)
- INC 1 ... INC 10000 (incremental jog)
- REFPOINT (Traverse to reference point)

For this purpose, use the operating mode selection key, and then the appropriate softkey (see below) or the operating mode selector switch (external machine control panel).

PART PROGR.



Now press the "PART PROGRAM" softkey.

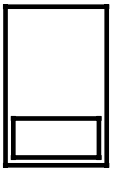
Apart from program input, the PART PROGRAM function also serves for:
- ***Program editing (see Section 3.1.9),***
- ***Program management (handling, overview, see Section 3.1.10)***
- ***and Program simulation (see Section 3.1.11).***

aus 0HQ01, S. 3-28
Änderungen s. Anlage

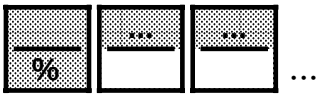
The display appears on the screen.

The control shows you the **last** part program called, in the example program number ” % 1”.

EDIT

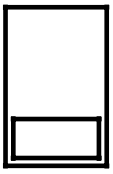


Press the ”EDIT” softkey.



Enter the desired program no. % **...** using the keyboard.

SELECT PROGRAM



Press the ”SELECT PROGRAM” softkey.

Now you can either input a new program or modify a part program that is already in the control using the input screen form displayed.

Example:

Entering an individual block

Enter the block number ”N ... ” using the numeric keys, and conclude with the input key.

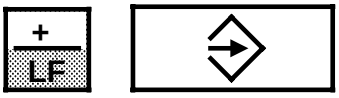
Enter the first word (in the example ”G01”) and conclude with the input key.

Enter the second word (in the example ”G90” for ”absolute”) and conclude with the input key.

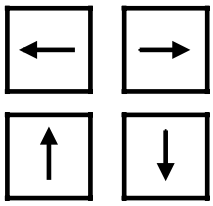
Enter, for example, a target coordinate (in the example”X25”) and conclude with the input key.

etc.

etc.



You end the block with ”LF” (block end) and conclude with the input key.



With these keys you can **position** the, cursor at any point in a stored part program; at the start of any block, or before any word within a block.

With this key you can **delete** either the complete block or a single word before which the cursor is positioned. The block number or word must have been entered in the input line.

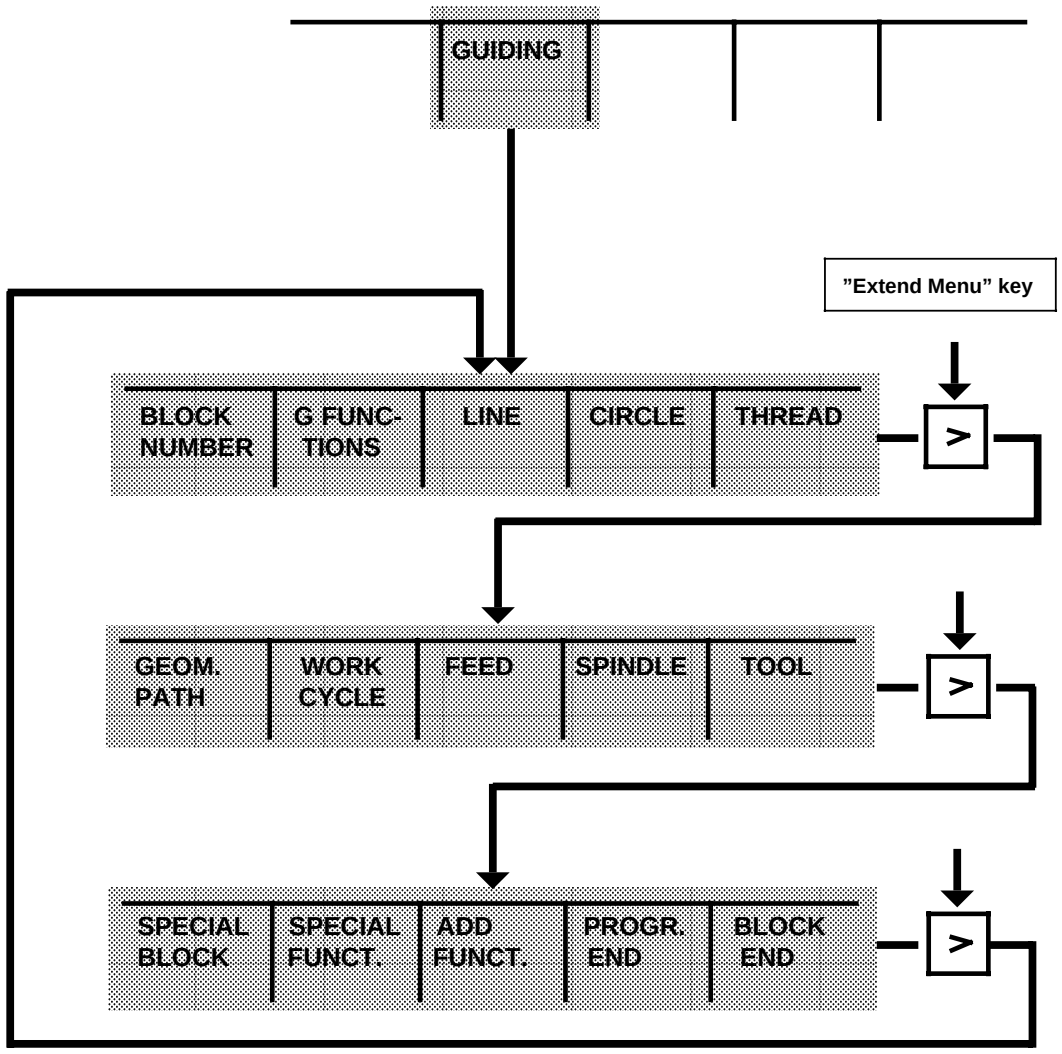
With this key you can **modify** either the complete block or a single word before which the cursor is positioned. The new word must have the same address character and have been entered in the input line.

3.1.7.2 Program input with operator guidance

Program input **with** operator guidance means:

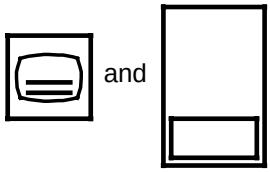
You enter an opened program % ... **with the aid of a menu** :
Frequently occurring input functions (e.g. G functions, M functions, complete contour paths etc.) are offered as menus. You can select and enter the desired function in this menu with the appropriate softkey.
Menu-supported input will save you time and operating work and reduces input errors.

To obtain the ”operator guiding” function, press the ”GUIDING” softkey. The softkey function shown below are then available.



Sequence of operation

You wish, for example, to enter block ”N5 G90 G00 X25 LF” into program no. ”%12”:

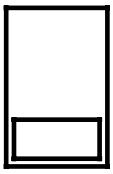


Select one of the following operating modes:

- PRESET (Actual value setting)
- JOG (Feed/jog)
- AUTOMATIC (Automatic mode)
- INC 1 ... INC 10000 (Incremental jog)
- REFPOINT (Approach reference point)

To do that, use the operating mode selector key, and then the appropriate softkey (see below)
or the operating mode selector switch (external machine control panel)

PART PROGR.



Now press the ”PART PROGRAM” softkey.

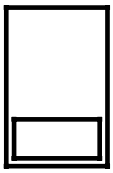
Apart from program input, the ”PART PROGRAM ”function also serves for:

- ***Program editing (see Section 3.1.9),***
- ***Program management (handling, overview see Section 3.1.10) and***
- ***Program simulation (see Section 3.1.11).***

aus 0HQ01, S. 3-32
Änderungen s. Anlage

This display appears on the CRT.
The control shows you the **last** part program called, in the example program no. ”%1”.

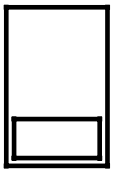
EDIT



Press the ”EDIT” softkey.

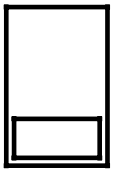
Enter the program number ”%12” using the keyboard.

SELECT PROGR.



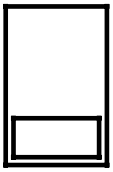
Using the ”SELECT PROGRAM” softkey, call the program.

GUIDING



Press the ”GUIDING” softkey.

BLOCK NUMBER

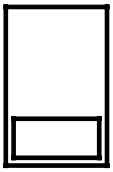


Press the ”BLOCK NUMBER” softkey.
Block number **”N0005”** is displayed.

Note:

The block numbers are generated by the control in steps of five, i.e. after each LF (end of block) the block number is incremented by **”5”**. After a scroll lock with ”+” and ”input”, the block number is not displayed.

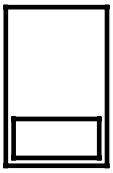
**G FUNC-
TIONS**



Press the ”G FUNCTIONS” softkey.

Extend the softkey menu until the required G function appears.

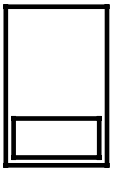
G90



By pressing the ”G90” softkey,
select the first G function required.
It is **not necessary** to press the input key.
”G90” is displayed.

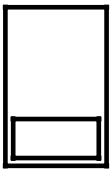
Press this key to return to a higher-level softkey menu.

LINE

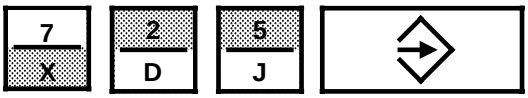


With the ”LINE”softkey, select
”linear interpolation”.

G00



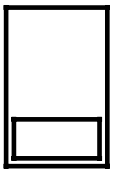
By pressing the ”G00” softkey,
select the ”Rapid traverse” function.
”G00” is displayed.



Using the numeric keys, enter the desired ”X” coordinate, with the value ”25”, and store it with the input key.

”**X25**” is displayed.

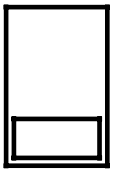
BLOCK END



Complete the block by pressing the ”BLOCK END” softkey.

”**LF**” is displayed.
The complete block is transferred into the part program memory.

BLOCK NUMBER



Press the ”BLOCK NUMBER” softkey. The control generates the next number, ”5” higher.

”N0010” is displayed and transferred into the part program memory.

etc. etc.

Continue until the part program is complete.

3.1.8 Contour elements

The ”CONTOUR” softkey function is an extension of ”Program input with operator guidance” (Section 3.1.7.2).

With ”CONTOUR” you can select contour elements. Graphics displays support the programming of specific geometric contour values.

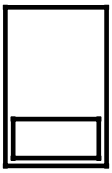
The required displays must be stored in the user memory submodule.

Sequence of operation

Select one of the operating modes as described in 3. 1. 7. 1 or 3. 1. 7. 2:

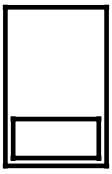
- PRESET
- JOG
- AUTOMATIC
- INC 1 ... INC 10000
- REFPOINT.

PART PROGR.



Now press the ”PART PROGRAM” softkey.

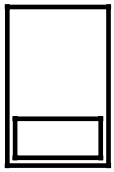
EDIT



Press the ”EDIT” softkey.

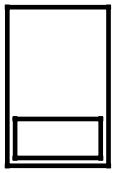
Use the numeric keys to enter the desired program number %... .

SELECT PROGR.



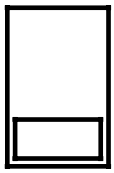
Call desired program using ”SELECT PROGRAM”.

GUIDING

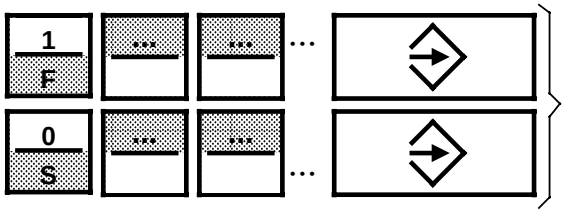


Press the ”GUIDING” softkey.

BLOCK NUMBER



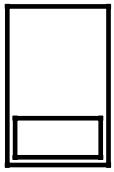
Press the ”BLOCK NUMBER” softkey. The control generates the lowest available block number N.. on the screen.



Enter the technology functions for feed **F**... and spindle **S...** using the numeric keys and store them using the input key.

Extend the softkey menu.

CONTOUR



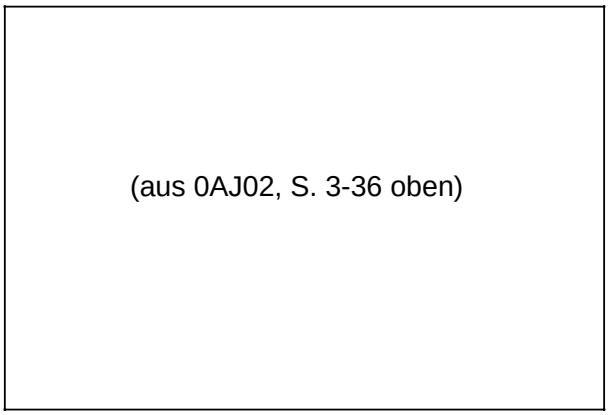
Select the ”CONTOUR” softkey.

(Bild aus 0AJ02, S. 3-35)

This display appears.

The available softkey functions are displayed in the menu.

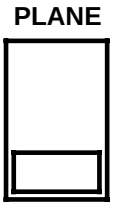
When you press the ”extend menu” key...



... this display appears with further softkey functions.

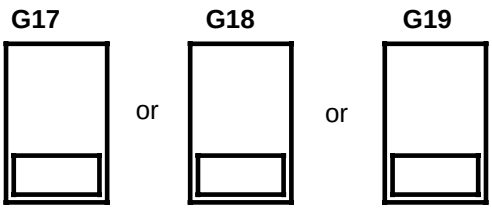
When you press the ”extend menu” key, the previous display reappears.

Before programming the contour, select the plane:



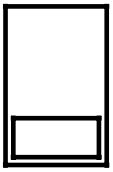
Press the ”PLANE” softkey

This display appears on the screen.

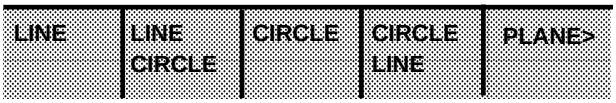


Select the plane you require with one of these softkeys

CONTOUR



Then press the ”CONTOUR” softkey in the same softkey menu.



The menu which then appears offers you these contour elements ...

... or extend the menu with this key ...

... and select one of the contour elements from this menu.

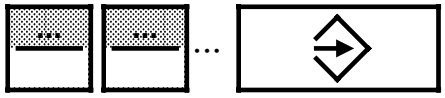
LINE CIRCLE

Now go back to the preceding display and select, say ”LINE CIRCLE” by pressing the appropriate softkey.

(aus 0HQ01, S. 3-38
Änderungen s. Anlage)

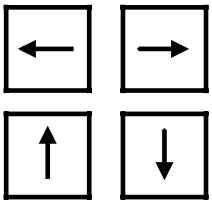
This input screen form appears with the input field marked in inverse video.

The contour element selected is displayed graphically.

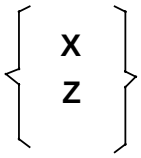


Now enter the first value using the numeric keys; then store this value using the input key.
The inverse video marking now jumps to the next input field of the contour: the control now waits for your next input...etc.

Now enter, one after the other the rest of the geometry values into all of the input fields of the contour marked in inverse video as described above.



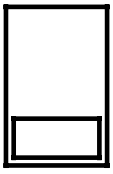
Using the cursor keys, you can position the inverse video marking to any input field, if, for example, you want to modify the geometry values for the selected contour.



Note:

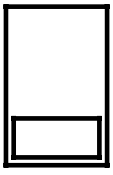
Note the input fields marked in inverse video on the CRT display, in curly brackets, in which there are several values (”X” **and** ”Z” in the example).
In this case you are only allowed to enter **one** value (i.e. only the X value **or** the Z value, in the example).

DELETE PARAM.



If you have inadvertently entered two values, press the ”DELETE PARAMETER” softkey.

STORE



When you have entered all the values required for the contour, press the ”STORE” softkey.

The contour is now stored in the part program with all the entered geometry values. The control generates the ”end of block” (LF) character itself, and shows you the complete block entered.

3.1.9 Program editing ("EDIT"): insert/modify/delete word, insert/delete block

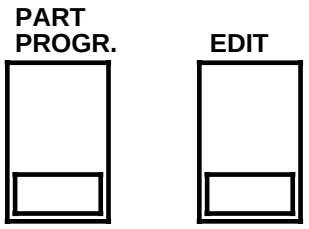
You can edit any part program stored in the program memory.

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

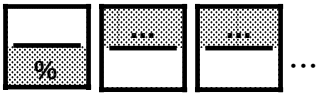
The following editing functions are possible:

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

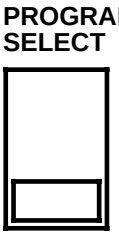
Sequence of operation



Press the "PART PROGRAM" and then the "EDIT" softkeys in any of the operating modes : PRESET/JOG/AUTOMATIC/INC1 ... INC10000/REFPOINT.

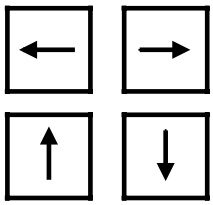


Using the keyboard, enter the %... (or "L...") number of the program you want to edit.



Call the desired program using the "PROGRAM SELECT" softkey.

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

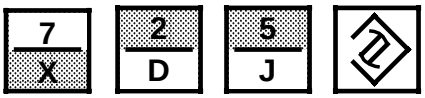


Using the cursor keys, set the cursor, marked in inverse video, in front of the point to be edited.

Insert word

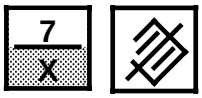
Enter the address **and** the new value ("X15" in the example) and store it using the input key.

Modify word



Enter the address of word **and** the **new** value ("X25" in example) and press the "modify word" key.

Delete word

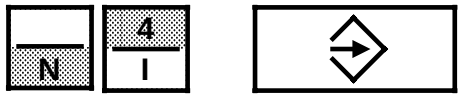


Enter the address of word to be deleted ("X" in the example) **and** press the "delete word/block" key.

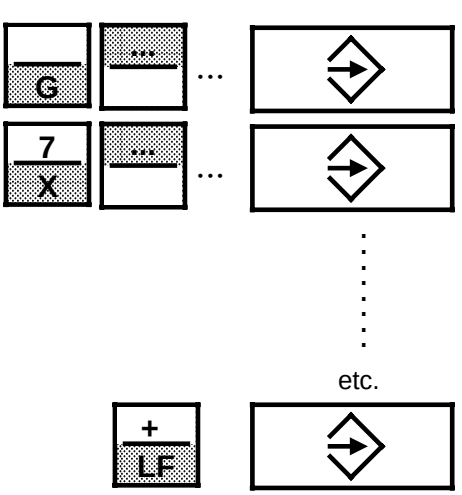
Insert block

N 5 .....LF

Position the cursor **in front of** the block which will **follow** the inserted block, (in the example a new block will be inserted in front of block **"N5"**).



Enter the block number to be input ("N4" in the example) and store it with the input key.



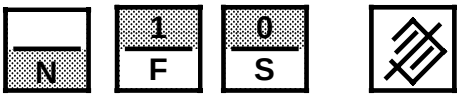
Using the keyboard now enter the block information word for word, and store each time with the input key.

End the new block with "LF" (end of block), and also complete this input with the input key.

Delete block

■ N10 ……LF

Position the cursor ■ **in front of** the block to be deleted ("N10" in the example).

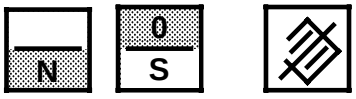


Using the keyboard, enter "N10" and press the "delete word/block" key.

For blocks without a block No. N... proceed as follows:

■ G …..F…..X…..LF

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



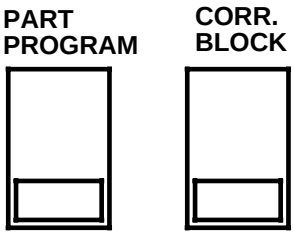
Using the numeric keys, enter "N0" (0=zero) and then press the "delete word/block" key.

With these key strokes, you can delete a program block generated by an OGM.

3.1.9.1 Debugging (”CORRECTION BLOCK”)

If the control detects a programming error while machining or simulating (3.1.11), execution of the program is stopped and the appropriate alarm is displayed. If you then press ”CORRECTION BLOCK”, the block or the word with the error is marked and can be edited.

Sequence of operation



Press the ”PART PROGRAM” softkey, and then the ”CORRECTION BLOCK” softkey, in any of the operating modes: PRESET/JOG/AUTOMATIC/
INC 1 ... INC 10000/REFPOINT.

The cursor is now positioned in front of the block/word recognised as containing an error.

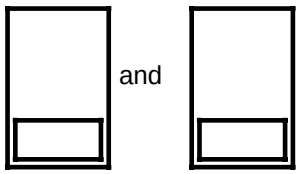
Using the keyboard, now correct the relevant address (address ”X” in the example) and enter the appropriate value:
Then press the ”modify word” key.

Having corrected the error, press the ”program start” key to continue program execution.

3.1.10 Program management

3.1.10.1 Displaying the stored programs (”DIRECTORY”)

Sequence of operation

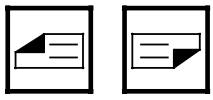


Press the ”PART PROGRAM”, and ”DIRECTORY” softkeys, one after the other in any of the operating modes: PRESET/JOG/AUTOMATIC/ INC 1 ... INC 10000/REFPOINT.

aus 0HQ01, S 3-44
Änderungen s. Anlage

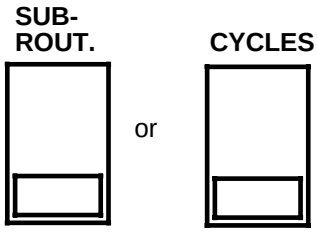
This relevant directory is displayed.

All main programs are listed.



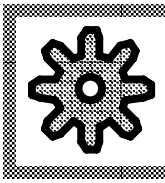
When the display is full :

You can press the”page forwards/backward” keys to obtain displays of directories of further stored main programs.

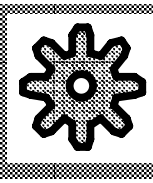


The displays of the directories for: ”SUBROUTINE” or ”CYCLES” are selected using the softkey menu on the CRT.

”Page up/down ” is possible as described previously.

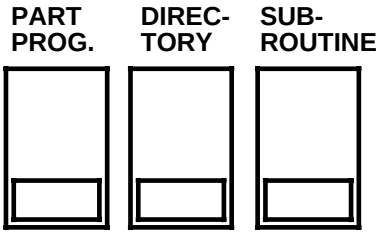
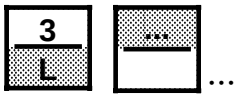
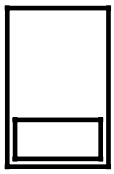


The programs can be displayed in the form of a column with the program comments to the right of it, if set in the setting data.



3.1.10.2 Protection of subroutines (cycle lock)

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

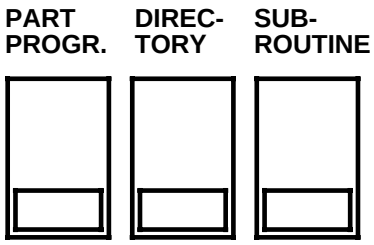
Sequence of operation	
	Press the ”PART PROGRAM”, ”DIRECTORY” and ”SUBROUTINE” softkeys, one after the other, in any of the operating modes: PRESET/JOG/AUTOMATIC/INC 1...INC 10000/ REFPOINT.
	Using the keyboard, enter ”L” for subroutine, and the number of the subroutine.
	Now press the ”LOCK” softkey [1].

The protected subroutine is entered in the cycles list. Cycles are protected subroutines.

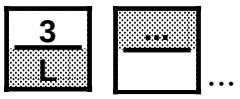
1) Only possible when cycle lock is not active.

3.1.10.3 Deactivate cycle lock (”UNLOCK”)

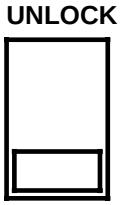
Sequence of operation



Press the ”PART PROGRAM”, ”DIRECTORY” and ”SUBROUTINE” softkeys, one after the other, in any of the operating modes: PRESET/JOG/AUTOMATIC/INC 1...INC 10000/ REFPOINT.



Using the keyboard, enter ”L” for subroutine, and the number of the subroutine.



Now press the ”UNLOCK” softkey.

The ”cycle” is entered in the list of (unprotected) subroutines.

Whether the cycle lock can be deactivated for cycles stored on the EPROM depends on the PLC program. The cycles can be protected against unauthorized output by programming the PLC accordingly!

This effectively protects the machine tool manufacturer's or user's know-how.

It is then no longer possible for the operator to lock and unlock programs at will!

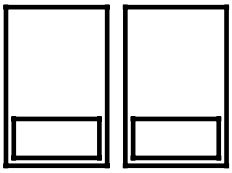
3.1.10.4 Copying a program (”COPY”)

Using the ”COPY” function, you can store the same program under a different program number in the program memory.

This enables you to keep one program with the contents unchanged and to make changes to the copy.

Sequence of operation

PART PROGR. **PROGR. HANDLING**



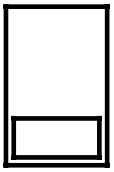
In any of the operating modes:
PRESET/JOG/AUTOMATIC/INC 1 ... INC 10000/
REFPOINT, press the ”PART PROGRAM”, and ”PROGRAM HANDLING” softkeys one after the other.

(aus 0HQ01, 3-47
Änderungen s. Anlage)

This CRT display is shown, - program number ”%1” in the example, the last program to be called.

Using the keyboard, enter:
” %1= %5 ”

COPY



Now press the ”COPY” softkey.

The control now copies program ”%1” and re-stores it under program number ”%5”.

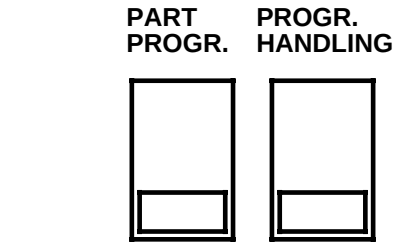
Program ”%1” ” remains unchanged!

3.1.10.5 Moving a program (”MOVE”)

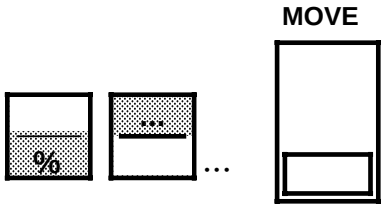
Using the ”MOVE” function, you can move the selected program to the end of the memory.

Only the program at the end of the memory can be modified while another program is being executed.

Sequence of operation



In any of the operating modes: PRESET/JOG/AUTOMATIC/INC 1 ... INC 10000/ REFPOINT, press the ”PART PROGRAM”, and ”PROGRAM HANDLING” softkeys one after the other.



Using the keyboard, enter the desired program number, ”%...” (or ”L...”), and press the ”MOVE” softkey.

The program selected is moved to the end of the memory.

Note:

If the cycle lock has been deactivated by the PLC, you can use the ”MOVE” function to copy a cycle from the EPROM to the end of the program memory. You can then edit the cycle in the same way as a normal part program.

As the cycle now exists twice with the same number L ... (in the EPROM and the program memory), the cycle in the EPROM is marked (*) in the directory with an asterisk. The cycle in the program memory has priority on selection.
The cycle in the EPROM can only be called again using the ”select program” softkey when the cycle in the program memory has been deleted.

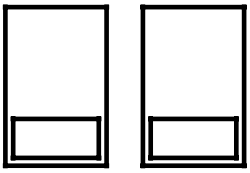
3.1.10.6 Renaming a program (”RENAME”)

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

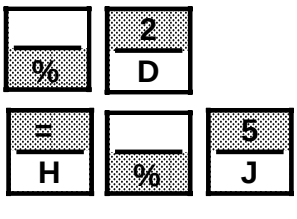
The contents of the renamed part program remain unchanged.

Sequence of operation

PART PROGR. **PROGR. HANDLING**

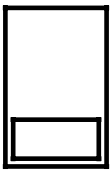


In any of the operating modes: PRESET/JOG/AUTOMATIC/INC 1 ... INC 10000/ REFPOINT, press the ”PART PROGRAM”, and ”PROGRAM HANDLING” softkeys one after the other.



Using the keyboard, enter the old program number (”%2” in the example), press the ”=” key, and enter the new program number (”%5” in the example).

RENAME



Press the ”RENAME” softkey.

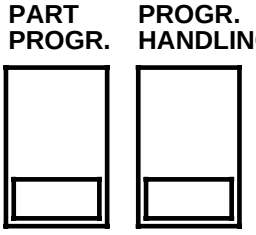
The (former) program ”%2” now has the program number ”%5”.

The ”RENAME” and ”DELETE” softkeys are positioned next to each other! You could inadvertently delete a program by pressing ”DELETE”.

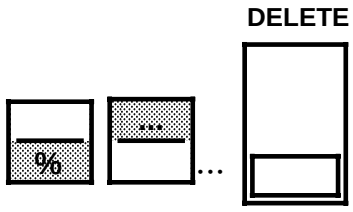
3.1.10.7 Deleting a program (”DELETE”)

Using the ”DELETE” function, you can either delete each program individually or several at a time.

Sequence of operation



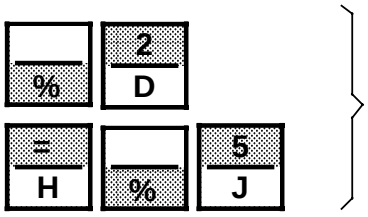
In any of the operating modes: PRESET/JOG/AUTOMATIC/INC 1 ... INC 10000/ REFPOINT, press the ”PART PROGRAM” and ”PROGRAM HANDLING” softkeys one after the other.



Deleting an individual program:

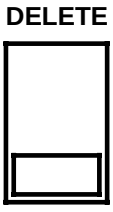
Using the keyboard, enter the desired program number ”%...” (or ”L...”), and press the ”DELETE” softkey.

The program entered is deleted by the control.



Deleting a sequence of programs

Using the keyboard, enter for example ”%2”, then press the ”=” key, and enter, for example, ”%5”.



Operate the ”DELETE” softkey.

All programs from ”%2” to ”%5” inclusive are now deleted by the control!

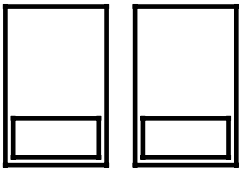
In AUTOMATIC mode, you can delete a program while it is running.

3.1.10.8 Reorganizing the program memory (”REORG”)

Using the ”REORG” function, you can reorganize the contents of the program memory.
The space that has been cleared by deleting the programs, can be used again for the input of programs.

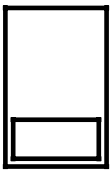
Sequence of operation

PART PROGR. **PROGRAM HANDL.**



In any of the operating modes:
PRESET/JOG/AUTOMATIC/INC 1 ... INC 10000/
REFPOINT, press the ”PART PROGRAM” and ”PROGRAM HANDLING” softkeys one after the other.

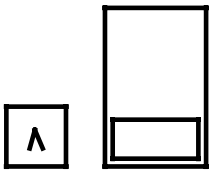
REORG



Press the ”REORG” softkey.

The program memory is reorganized by the control.

DIRECT-ORY



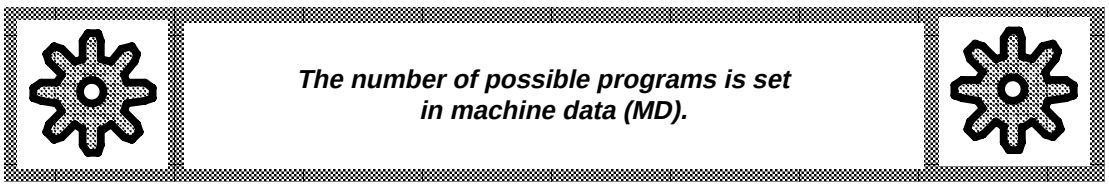
Switch back to the higher-level menu for softkey functions, and press the ”DIRECTORY” softkey.

The available memory space is displayed on the CRT.

Attention:

The available (displayed) memory cannot be used to its full capacity !
The following must be deducted from the displayed capacity:

- 10% are reserved gaps (gaps are provided for subsequent editing)
- plus: 11 characters multiplied by the number of possible programs!

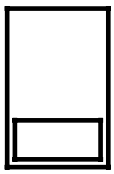


3.1.10.9 Selecting a program

Using ”SELECT PROGRAM”, you can either start a new program, or call a program already stored in the program memory, e.g. for editing (3.1.9) or simulation (3.1.11) .

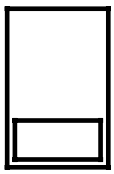
Sequence of operation

PART PROGR.



Press the ”PART PROGRAM” softkey in any of the operating modes:
PRESET/JOG/AUTOMATIC/INC 1 ... INC 10000/ REFPOINT.

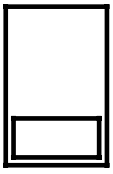
EDIT



Press the ”EDIT” softkey.

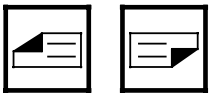
Using the keyboard, enter the program number ”%...” (or ”L...”), ”%5” in this example.

SELECT PROGRAM



Press the ”SELECT PROGRAM” softkey.

The program number called is shown on the CRT and, if there is a program with this number in the memory, the start of the program and blocks N0005 to N ....



Using the ”page up/down ” keys, you can call up the parts of the program not yet displayed (if in memory) on the CRT.

3.1.10.10 Operator guidance macros (OGM) and back translation

Precondition:

Input displays and the associated operator guidance macros developed by the machine manufacturer on the WS800A configuration system are stored on the UMS of the system and can be activated. The displays shown in this section are only examples. See the machine manufacturer's instructions for current menus, display descriptions and operating sequences.

Assignment of parameters to part programs can be supported graphically using these configured input displays. If the operator inputs values into a display of this type, they are stored in the configured input buffer (MIB) when the INPUT key is pressed. These values can be inserted into the program section preconfigured by the operator guidance macro (OGM) and placed in the part program in the program memory using the softkey function "store".

Parts and data blocks of this kind can be present in a part program several times with the same structure but variable values.

The function "back translation" permits you to display a part program section generated using OGMs in its input display again, then to display the data and update the program with the softkey function "store".

Operator guidance macros and configured input displays are developed using WS 800A. The result of such configuration might be:

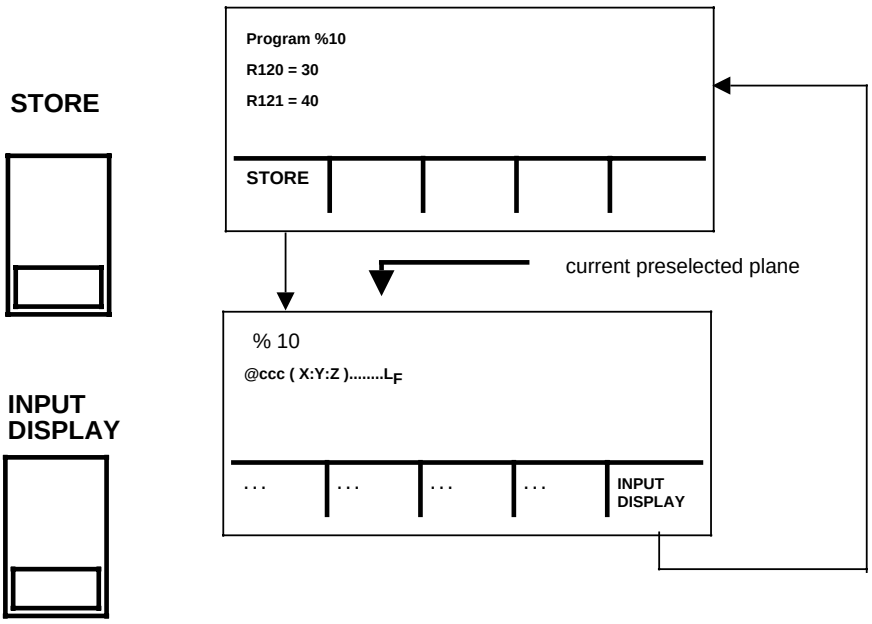
```
%BFM 7
(OGM: plunge-cut)
R1= ~101 R12= ~112 ...
```

The OGM number is between 1 and 998.
%BFM 999 contains are cross-reference list of all OGMs.

e. g.

```
%BFM 999
Plunge-cut = 7.34  <name> = <OGM No.>, <MB No.>
Dressing = 8.35  MB=menu block
```


Example of a data block



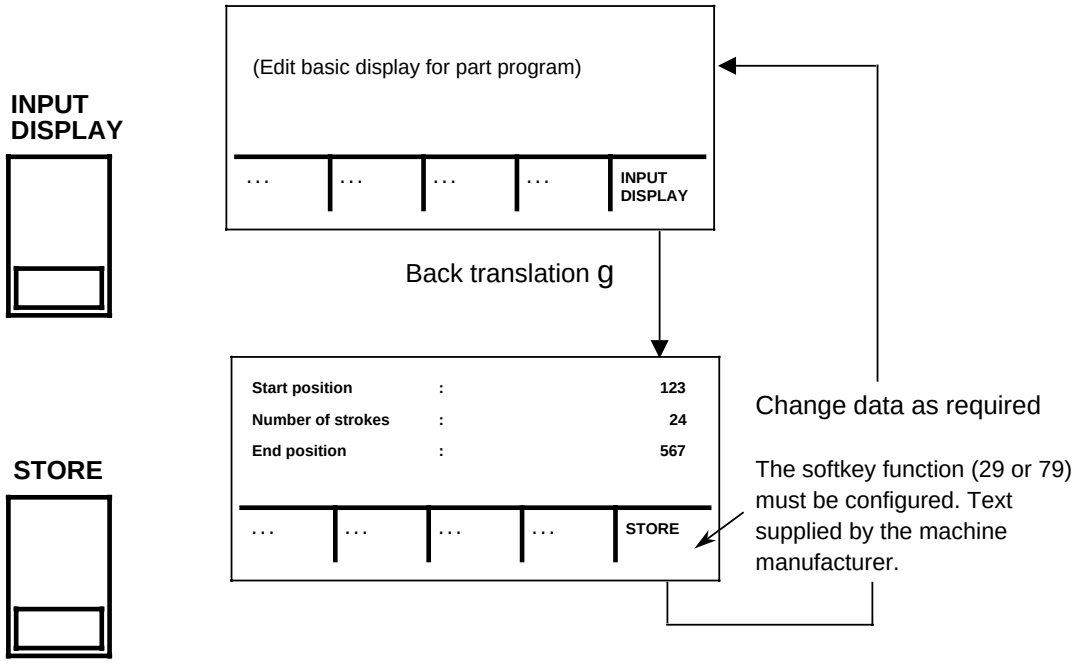
Meaning of the parenthesis expression (X:Y:Z):
The content of the parenthesis expression represents the current plane selection as follows:

X : Y : Z abscissa : ordinate : applicate .

The plane selection must be compatible with the following contour definition. I.e. if the plane selection has been changed using the function EDIT, back translation of the data block might not be possible.

Back translation

Move the cursor into the required data block in the basic display part program under EDIT and press the softkey INPUT DISPLAY.



With "store", the changed values are transferred to the appropriate data block in the part program. The basic display part program is displayed again.

DIN instructions can be inserted between the data blocks generated by OGMs. Insertions within a data block make back translation of that data block impossible.

Example of the result of a part program

```
 %1234                                          if        BFM:
 :                                                       %BFM 7
 @ccc (X:Z:Y) N10 LF                                     (OGM: plunge-cut) N 100
R11=123 R12=24 . . . (OGM: plunge-cut) LF                R11= 101  R12= 112
```

or

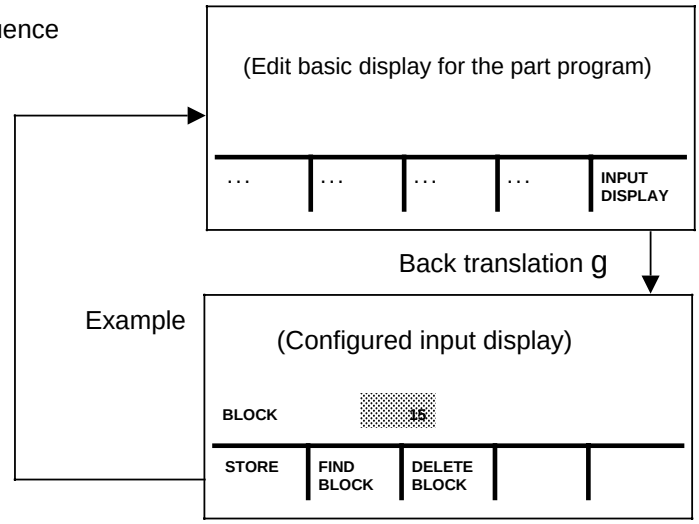
```
%1234                                           if       %BFM 7
  :                                                      (OGM: plunge-cut) N 100
@ccc (X:Y:Z)  N10 LF                                    R11 =  101
R11 = 123 LF                                            R12 =  112
R12 = 24 LF                                                :
  :                                                        :
(OGM: plunge-cut)  LF
```


Function FIND BLOCK using block number

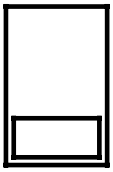
Precondition:
The configured input display must be prepared.

e.g. in the OGM	in the program block
%BFM7	**%1234**
(OGM: SCRATCHING) N~100	**: @ccc (X:Y:Z) N15**
:	**:**
:	

Operating sequence

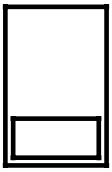


FIND BLOCK



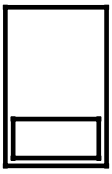
Current values are transferred into the specified input fields. The data block is selected via the block number field (BLOCK in the example).

STORE



Data transfer into the program block

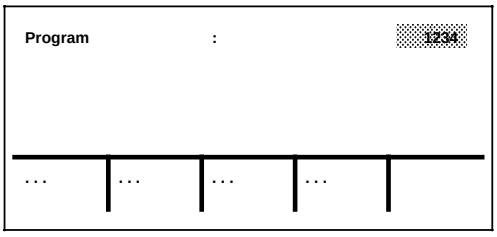
DELETE BLOCK



With this key, you can delete the data block with the preselected number can be deleted.

Function PROGRAM PRESELECTION via a configured display

Precondition;
The configured display must be selected for this function.



Configured display

Example

The program number entered is preselected in the display for part program editing (EDIT).

3.1.11 Simulation

With the ”SIMULATION” function, you can test a program. The traversing movements of the tools are simulated graphically on the CRT. The functions G05 and G07 cannot be simulated in the 3rd channel.

3.1.11.1 Simulation area of the workpiece

- A graphic display with input fields simplifies the input of the values for the simulation area and the dimensions of the workpiece.
- Select the simulation area so that the programmed traversing range is covered.
- The values entered need not be in proportion to the display; the simulation area is not cut off, because the control automatically makes the display big enough for the largest value.

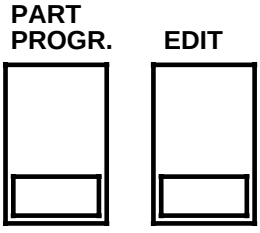
The machining channels and the simulation channel can influence each other via common data.

Common data include:

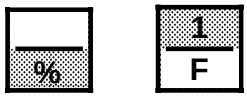
- Settable zero offsets
- Programmable zero offsets
- Tool offsets
- Global R parameters
- Machine data
- Setting data.

Any influencing of the data of the machining channels by the simulation channel can be prevented by an interlock. Simulation simultaneous with machining is then not possible!

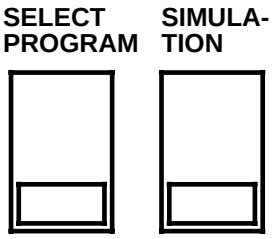
Sequence of operation



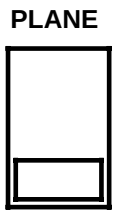
In any of the operating modes:
PRESET/JOG/AUTOMATIC/INC 1 ... INC10000/ REFPOINT, press the ”PART PROGRAM” and ”EDIT” softkeys one after the other.



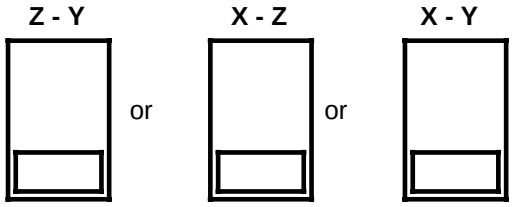
Using the keyboard, enter the program number "%..." (or "L..."), ("%1" in this example).



Press the "SELECT PROGRAM" softkey, and then the "SIMULATION" softkey.



Press the "PLANE" softkey.

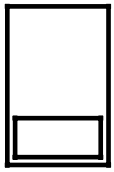


Select the plane with one of these softkeys.

(aus 0AJ02, S. 3-53)

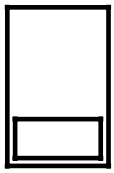
The simulation plane you have selected is marked in the display in inverse video.

**SIMULA-
TION**



Press the ”SIMULATION” softkey.

**AREA
W-PIECE**

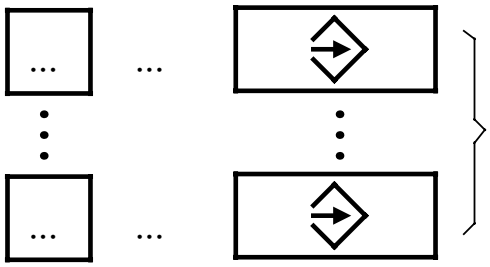


Now select the ”AREA W-PIECE” function in the newly displayed softkey menu.

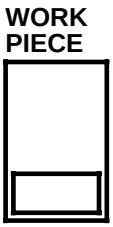
(aus 0AJ02, S. 3-54)

This input form with the input fields for ”simulation area” appears.

The current input field, into which you can input values, is displayed in inverse video.



Using the keyboard, now enter the appropriate **values** for the ”simulation area”. Terminate each input with the input key. After each input, the inverse video marking jumps to the next field.

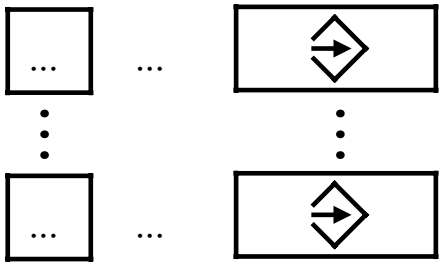


Select the ”WORKPIECE” function with the appropriate softkey.

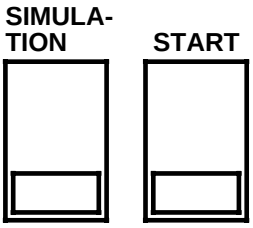
(aus 0AJ02, S. 3-55)

This input screen form appears with the input fields for the workpiece dimensions.

The current input field, into which you can input values, is identified in inverse video.



Using the keyboard, enter the appropriate dimension for ”workpiece” into the input form. Terminate each input with the input key.
After each input, the inverse video marking jumps to the next field.



Press the ”SIMULATION” and ”START” softkeys, one after the other.

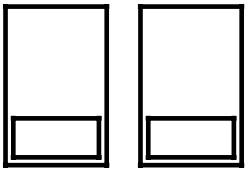
The control now simulates the programmed motion on the CRT.

3.1.11.2 Influencing simulation

The simulation can be influenced so that it is adapted to the program sequence in the ”AUTOMATIC” mode.

Sequence of operation

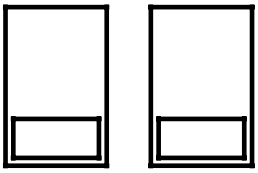
PART PROGR. **EDIT**



Press the ”PART PROGRAM” and ”EDIT” softkey, one after the other in any of the operating modes: PRESET/JOG/AUTOMATIC/INC FEED 1 ... INC FEED 10000/REFPOINT.

Using the numeric keyboard, enter the program number ”% ...” (or ”L...”), (”% 1” in the example).

SELECT PROGRAM **SIMULA-TION**



Press the ”SELECT PROGRAM” and ”SIMULATION” softkeys one after the other.

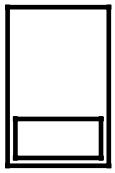
Using this key, extend the displayed softkey menu.

The new softkey menu gives you four options:

PROGR. CONTROL	BLOCK SEARCH	DISPLAY TYPE	SIMUL.-DATA
a)	b)	c)	d)

a) ”PROGRAM CONTROL”

PROGRAM CONTROL

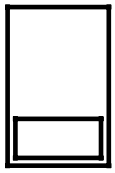


When you press the ”PROGRAM CONTROL” softkey, ...

(aus 0AJ02, S. 3-57 oben)

... this CRT display is shown. You can alter the simulation ”status” using a softkey, by selecting the desired function and setting it to ”YES”.

SIMULA-TION

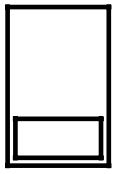


>

Now press the ”SIMULATION” softkey again, and then press the key to extend the displayed softkey menu.

b) ”BLOCK SEARCH”

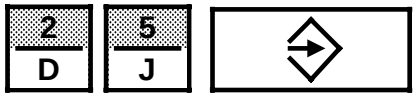
BLOCK SEARCH



Using the softkey, select the ”BLOCK SEARCH” function.

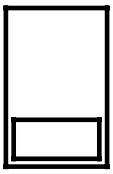
(aus 0AJ02, S. 3-57 unten
Änderungen s. Anlage)

This display is shown on the CRT.



Using the keyboard, enter the desired block number ”N...” (”25” in the example; ”N” need not be entered) and store it with the input key.

START

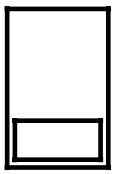


Press the ”START” softkey.

Simulation does not start at the beginning of the program but at the block selected (N25 in the example).

c) ”DISPLAY TYPE”

DISPLAY TYPE



Press the ”DISPLAY TYPE” softkey.

(aus 0AJ02, S. 3-58)

You can select one of two types of display: ”CONTOUR” or ”ERASE”.

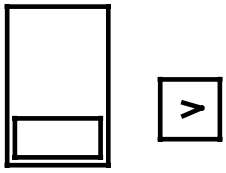
Select the one you require with the appropriate softkey.

Note:

”ERASE CENTRE” is an option.

"ERASE CENTRE" means: "erase with display of the centre path of the grinding tool".

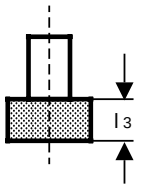
SIMULA-TION



Now press the "SIMULATION" softkey again and then the key to extend the softkey menu displayed. This takes you back to the softkey menu for simulation control.

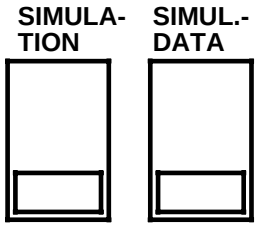
Note:

For the simulation you can enter the length l_3 for three tool offset numbers D ... (this is not taken into account in the tool data (3.1.4)):



The length l_3 is visible in the plane concerned when ”ERASE” is selected.

d) ”SIMULATION DATA”



If you press the ”SIMULATION” and ”SIMULATION DATA” softkeys after you have chosen the type of display ...

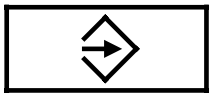
(aus 0AJ02, S. 3-59)

... this input screen form for ”simulation data” appears.

The current input field, into which you can enter/modify values, is marked in inverse video.

Now you can enter/modify up to three offset numbers D ... and the associated values for the length (l_3) using the keyboard.

Once you have made an input and pressed the input key the next input field is marked.



Example of the simulation of a part program (" %1 "):

```
%1
N1 G1 F100 Z50  LF
N2 X20 Y10 D3 LF
N3 Z10 LF
N4 L1 P5 LF
N5 M30 LF
```

Subroutine L1

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

aus 0HQ01, S. 3-66
Änderungen s. Anlage

The following display appears:

%1	Part program no.
N . .	Block number
L . .	Subroutine no.
P . .	Subroutine passes
X . ., Y. ., Z . .,	Current axis
C1 . ., C2 . .	values
X124.60	Simulation area for "X"
Y70.000	Simulation area for "Y"
F . .	Current path feedrate
F . .	Current simultaneous feedrate

T, D, H, functions are not displayed.
Subroutine nesting is simulated, but not displayed!

3.1.12 Data input/data output

For the input and output of data you will find a RS232 (V.24)/20 mA Interface (=**Interface 1**, with variable device connection) on the front of the operating panel.
A further V.24 (RS232) Interface (=**Interface 2**, with fixed assignment for the connection of a particular device) is located at the back of the control.

The second RS232C (V.24) Interface is only available when option "C62" has been ordered!

Fixed interface data are defined for the interfaces. These data can be modified by calling the "SETTING DATA" function.

Modification of the interface data - especially for interface No. 1 - is, for example, necessary when a different device is interfaced to the SINUMERIK 810G (printer, tape reader, programmer, etc.)

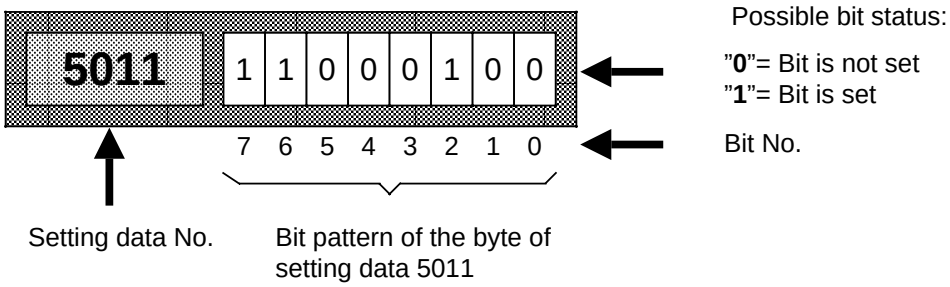
You will find the necessary information for setting/modifying the setting data bits in Section 3.1.12.1 (Operating sequences) and Section 6.3 (Setting data defaults for various connecting devices).

3.1.12.1 Setting data bits

The setting data - from byte No. 5010 (to 5029) - each consist of 8 bits: No. 0 to No. 7. These setting data are used to set interfaces 1 and 2 for the connection of different devices.

Explanation of setting data structure:

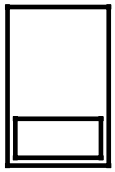
e.g. for setting data (SD) 5011



Using an input form on the CRT, you can enter/modify the bit pattern for any byte in the range **5010** to **5029**.

Sequence of operation

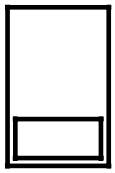
SETTING DATA



Press the ”SETTING DATA” softkey in any of the operating modes.

Using this key, extend the displayed softkey menu.

SETTING BITS

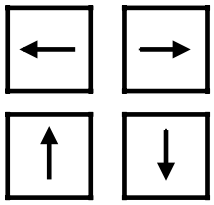


Using this softkey, select the ”SETTING BITS” function.

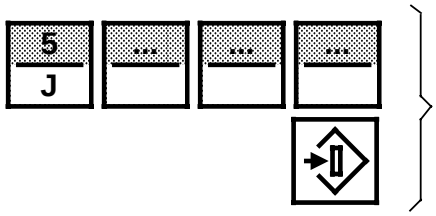
aus 0AJ02, S. 3-61

This input form appears.

The setting data in the top line is marked in inverse video.



Using the cursor keys, you can position the cursor on any other setting data or **...**



... using the keyboard, enter the number of the required setting data, and press the search key.

The input field marked in inverse video is now positioned on the 8 bit pattern of setting data number 5 **...** called.

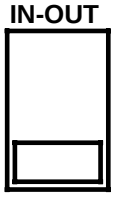
Using the ”0” or ”1” numeric keys, enter the desired bit pattern, and store the entire sequence of characters using the input key.

Carry on in this way until all the required setting data have been set.

3.1.12.2 Data input

Sequence of operation

DATA IN-OUT

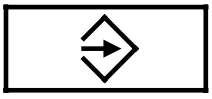


Press the ”DATA IN-OUT” softkey in any of the operating modes:
PRESET/JOG/AUTOMATIC/INC 1 ... INC 10000/ REFPOINT.

aus 0AJ02, S. 3-63
Änderung s. Anlage

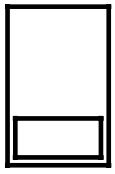
This input form is shown, with the input field for ”interface for input” marked in inverse video.

1
F
or
2
D



Now enter:
”**1**” (for input into interface 1) or
”**2**” (for input into interface 2)
and store it with the input key.

DAT. IN START



Select the softkey function ”DATA IN START”.
By doing this, you enable the control's receive function.

On the upper right-hand side of the CRT the message **DIO** (data input/output) is displayed.

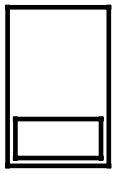
Under **Data type:** the data coming from the transmission device (tape reader, programmer ...) is identified.

The following table explains the types of data that can be loaded into the **810G:**

Data Type	Meaning
MPF	Part program (**M**ain **P**rogram **F**ile)
SPF	Subroutine (**S**ub **P**rogram **F**ile)
TOA	Tool offset (**T**ool **O**ffset **A**ctive)
ZOA 0	Zero offset (**Z**ero **O**ffset **A**ctive)
ZOA 1	K0... angle of rotation, channel 1
ZOA 2	K0... angle of rotation, channel 2
ZOA 3	Angle of coordinate rotation, channel 3
TEA 1	NC machine data (**Te**sting Data **A**ctive **1**)
TEA 2	PLC machine data (**Te**sting Data **A**ctive **2**)
PCA	PLC alarm text (**P**rogrammable **C**ontrol **A**larms)
PCP	PLC program (**P**rogrammable **C**ontrol **P**rogram)
RPA	Parameters with assigned values (**R P**arameter **A**ctive)
SEA	Addresses with assigned values (**Se**tting Data **A**ctive)
CLF	Clear instruction (**Cl**ear **F**ile)
ASM	**U**ser **M**emory **S**ubmodule

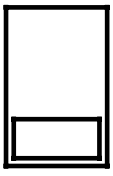
Under **Interface allocation:** the control gives you the **identifier** of the connected **device type** for interface 1 and interface 2, (for identification of various device types see Section 6.3).

STOP



Using the ”STOP” softkey, you can interrupt data input at any time.

DATA IN
START



Continues data input again.

During data input it is not possible for the control to call up specific data!

Notes:

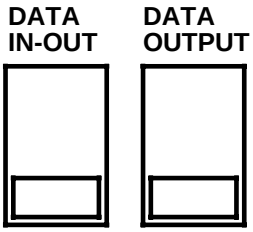
When inputting from punched tape, the input data are checked for simple errors.
All characters in punched tape code have a common characteristic:

- in ISO code there is always an even hole count
- in EIA code there is always an odd hole count.

The control automatically identifies whether the code is ISO or EIA by the first character, ”%” (ISO) or ”EOR” (EIA)!
The criterion ”odd hole count” or ”even hole count” is used to check the parity of each character from the second character in the program onwards.
As a further check, if a program already in the program memory is read in again, a complete program comparison is carried out.
When an error is detected, reading in is stopped and the error displayed on the CRT.

3.1.12.3 Data output

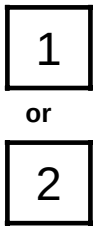
Sequence of operation

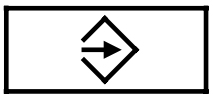


In any of the operation modes:
PRESET/JOG/AUTOMATIC/INC 1 ... INC 10000/ REFPOINT,
press the ”DATA IN-OUT” softkey and subsequently ”DATA OUTPUT”.

aus 0AJ02, S. 3-65
Änderung s. Anlage

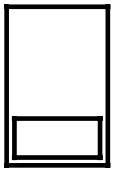
This input form, with the input field for ”interface for output” marked in inverse video, appears.





Now enter:
"**1**" (for output via interface 1) or
"**2**" (for output via interface 2)
and store it using the input key.

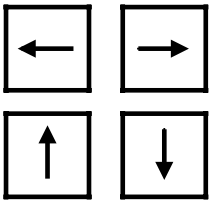
PART PROGR.



Now select, for example, the "PART PROGRAM" function with this softkey.

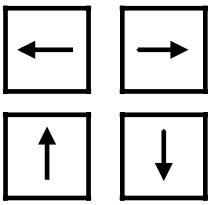
aus 0AJ02, S. 3-66
Änderung s. Anlage

This input form appears.

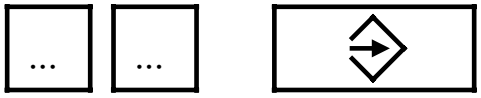


Using the cursor keys, now select the **upper** of the two main program "%..." lines or, to the right of it, the subroutine "L..." line.

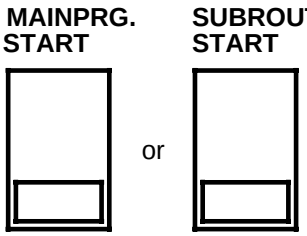
Using the numeric keyboard, enter the **number** of the main program, or the subroutine with which you want the output to **begin**.
Now store your input with the input key.



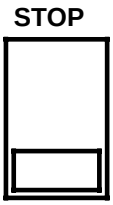
Again using the cursor keys, you select the **lower** of the two main program "%..." lines or, to the right of it, the subroutine "L..." line.



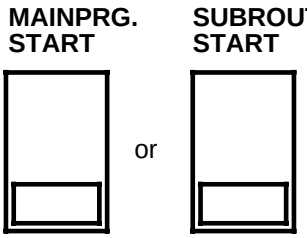
Using the numeric keyboard, enter the **number** of the main program, or the subroutine with which you want the output to **end**. Now store your input with the input key.



Press the
"MAIN PROGRAM START" or
"SUBROUTINE START" softkey.
By doing this, you start the control's transfer function (**output** of data).



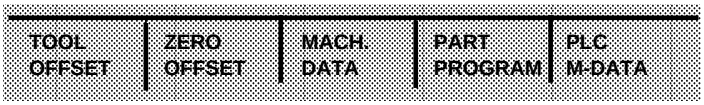
Using the "STOP" softkey, you can interrupt data output at any time.



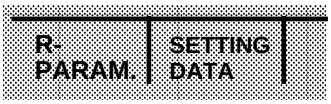
Start data output again.

After the transfer (data output) is complete, return to a higher-level softkey menu using this key.

You can now call any of these functions for data input (”PART PROGRAM” known - see above).

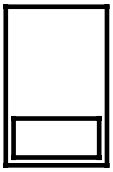


Using this key, you can extend the displayed softkey menu.



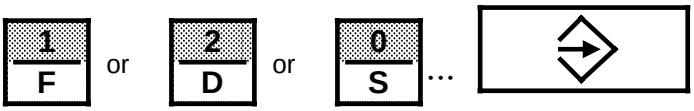
The displayed function gives you the option of outputting all the ”R parameters” and ”setting data”.

R- PARAM.



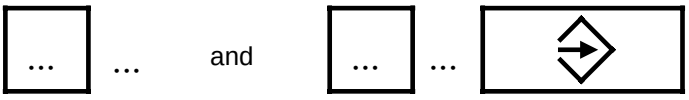
Press the ”R parameter” softkey.

This input screen form appears.



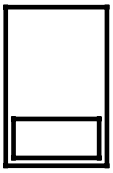
Using the keyboard, enter the channel No.:

”0” for output of the global R parameters
”1” for output of local R parameters for channel 1
”2” for output of local R parameters for channel 2
”3” for output of local R parameters for channel 3



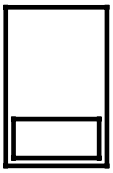
Using the keyboard, enter the beginning and the end of R parameter output.

START



Start the output of the R parameters by pressing the ”START” softkey.

STOP



Using this softkey, you can stop data output at any time...

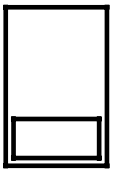
...and continue it again with ”START”.

In the same way, you can call and start the output of:

- ”TOOL OFFSET”
- ”ZERO OFFSET”
- ”MACHINE DATA”
- ”PLC MACHINE DATA”

not explained here.

ETX START



With the "ETX START" softkey you can output the ”end of transmission” character set in the setting data.

The character is always output regardless of whether the RTS line or XON/XOFF protocol is set.

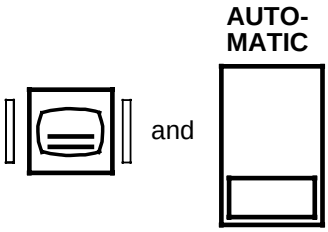
Note:

- A selection of the data, i.e. the output of parts of it, is not possible - as with ”PART PROGRAM”!
- You only need to enter a channel number when outputting ”R parameters”.
- Data output with or without leader and trailer can be set in the setting data (SD 5016, bit 1).

3.2 Machining

3.2.1 Starting a part program

Before starting a part program, the control and machine must be set up for the execution of the part program.



Press the operating mode selector key on the integrated machine control panel, and select the ”AUTOMATIC” softkey...

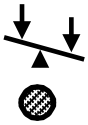
or 

... or rotate the operating mode selector switch on the external machine control panel to this symbol.

Enter the part program number, (”%77” in the example) - and store it with the input key.

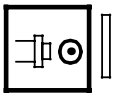
Press the ”program start” key on the machine control panel.

The green ”program running” LED display lights up.



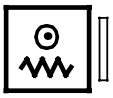
The green ”position not yet reached” LED display is bright until all axes have reached their set positions.

When the red ”feed hold” LED display is bright, program execution has been stopped!



Press the ”spindle start” key ...

or



... the ”feed start” key

By doing this, you enable program execution again, provided that **no program start disable** has been activated by the **PLC** (e.g. Alarm 2039).

After the program start, either ”ACTUAL (current) VALUES” or ”ACTUAL (current) BLOCK” appears on the CRT display.

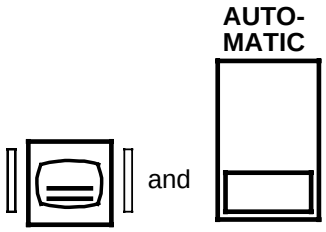
If, during machining with an active block continuing several feedrates, the keys NC STOP and NC START are pressed one after the other and no external bits are active, the feedrate F programmed in the block for the axes programmed in the block is used for further motion. The retraction path might not agree with the programmed value.

3.2.2 ”AUTOMATIC” mode- Selection of the ”ACTUAL (current) VALUES” or ”ACTUAL (current) BLOCK” display

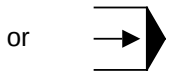
Precondition:

An active program is being processed in ”AUTOMATIC” mode.

Sequence of operation



Press the operating mode selector key in the integrated machine control panel, and then the ”AUTOMATIC” softkey ...



…. or rotate the operating mode selector switch on the external machine control panel to this symbol.

0HQ01, S. 3-79
Änderungen s. Anlage

The ”ACTUAL (current) VALUES” basic display for the active program is shown on the CRT.

Using this key, you can switch over to the second AUTOMATIC basic display.

0HQ01, S. 3-80
Änderungen s. Anlage

The second AUTOMATIC basic display. The meaning of the softkey functions change.

Using this key, you can return to the first automatic basic display!

Explanation of the second AUTOMATIC basic display

```
%77  N   0 L 0 N   0   P   0
```

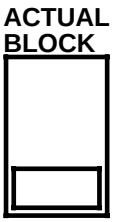
Display of the selected program (program pointer)
Execution pointer, with information on the block number ("N") and the subroutine number L..., number of passes P... and the block number within the subroutine with up to four levels of nesting.

```
Spindle      Set      Spindle  Actual
S1         0 U 0   S1       0   100%
S2         0 U 0   S2       0   100%
```

Setpoints and actual values of spindles S1 and S2.
Spindle speed in entry left of "rev" speed in rev/min
Spindle speed in entry right of "rev" constant grinding wheel surface velocity in m/sec

```
Actual Pos.  Feed      Set    Feed     Actual
             F 0.000M         F 0.000    85%
X   0.000    F 0.000M         F 0.000
Y   0.000    F 0.000M         F 0.000
Z   0.000    F 0.000M         F 0.000
Q   0.000    F 0.000M         F 0.000
```

Programmed and actual path feed.
Override setting in %. Current axis positions, programmed axis-specific feed values and current axis-specific feed value. Linear feedrates are marked with "M" and revolutional feedrates with "R".



Press the ”ACTUAL (current) BLOCK” softkey.

0HQ01, S. 3-81
Änderungen s. Anlage

The ”ACTUAL (current) BLOCK” display for the active program appears (program ”% 77” in this example).

Explanation of the "ACTUAL (current) BLOCK" display

Instead of the auxiliary functions and G functions (”ACTUAL (current) VALUES”), the following are displayed:

- The block **before** the ”current block”, here block ”N0005”
- The ”current block”, here block ” N0010”
- The block after the ”current block”, here block ”N0015”

Note:

If the ”current block” consists of more than 41 characters, the **following** block is **not** displayed.

Likewise, a block **before** or **after** the ”current block” with more than 41 characters is not displayed.

3.2.3 Influencing ”AUTOMATIC” mode

General notes:

The effect of influencing ”AUTOMATIC” mode (e.g. using programmed functions), is displayed in the first line of the CRT display.

Display: **STOP: AUTO interrupted**

Interruption of ”AUTOMATIC” mode (program stop),
see 3.2.3.4 ”program Interruption”

Display: **STOP: Single block**

A single block has been executed (with switch setting ”single block - ON”).

Display: **STOP: Pr. stop M00, M01**

Programmed interruption of program execution

with ”Program start”  execution of the program is continued.

Display: **STOP: Read enable**

Read enable is a PLC output signal. When the read enable signal is cancelled, execution of the current NC block is finished. The next program block is not transferred for processing.

Display: **STOP Dwell time**

Execution of the program is interrupted for the duration of the programmed dwell time.

Note:

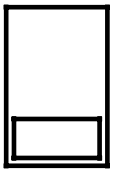
”Feed hold” is **not** displayed on the CRT!

3.2.3.1 Program control

When you press the softkeys described here, the active program is influenced.

Sequence of operation

PROGRAM CONTROL



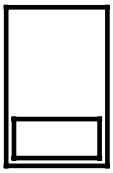
Press the ”PROGRAM CONTROL” softkey in the ”AUTOMATIC” operating mode.

aus 0AJ02, S.3-75

This CRT display with a menu of five functions appears.

By pressing any of the five softkeys, you can now change over the preset ”**NO**” to ”**YES**”. A function marked with ”**YES**” is **active**!

SKIP YES-NO



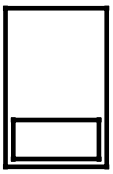
Selects, for example, the ”SKIP BLOCK” softkey function .

Display:

Skip Block	**YES**

Blocks in the program which are marked with an **oblique** before the block number (”/ N ...”) are skipped.

DRY
YES-NO



Selects, for example, the "DRY RUN FEEDRATE" softkey function.

Display:

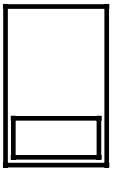
Dry run feedrate	**YES**

All blocks in which a feedrate is programmed (G01, G02, G03, G33, G34, G35), will now be traversed at the feedrate value entered into the "Dry run feedrate" setting data, instead of the programmed feedrate!

The dry run feedrate then applies to "feedrate per revolution" G95, and the feedrates for threading.

If the dry run feedrate is active, the reciprocation motion programmed with G181 is executed at the programmed feedrate.

OPT. STOP
YES-NO



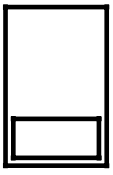
Selects, for example, the "OPTIONAL STOP" softkey function

Display:

Optional stop	**YES**

When an "M01" is present in the part program, the program will be stopped.
When the softkey function is marked with "**NO**", "M01" will be ignored.

OVERR.
YES-NO



Selects, for example, the "RAPID OVERRIDE" softkey function.

Display:

Rapid override	**YES**

This signal presupposes that there is only **one** common switch for both feedrate override and rapid traverse override. If the softkey function is set to **"NO"**, this override switch is only active with "feedrate".
If the softkey function is set to **"YES"**, the override set with this override switch is also active for rapid traverse.

User notes on ”RAPID OVERRIDE”:
When starting up a new PLC program the rapid override switch is unlocked with a keyswitch.

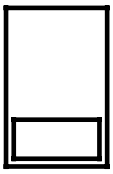
The following rapid override values can be set with the selector switch on the customer operator panel:

Setting	Override value in %
1	1
2	10
3	50
4	100

This table is only applicable when the ”rapid override switch present” machine data is set.

If this is not the case, the setting for the ”feedrate override switch” applies for rapid override, with a limitation to 100%!

DEC-SBL
YES-NO



Selects, for example, the ”DECODING SINGLE BLOCK” softkey function.

Display: **Decoding Single Block YES**

When the function is activated with ”**YES**”, the control generates the ”decoding single block active” signal.

The signal becomes active at the end of **the block** in the running part program, which is being **decoded** when the signal is set.

Now press the ”program start” key. The single block decoded is now executed.

Press the ”program start” key **once more**. The next single block decoded is executed ... etc.

Note:

The "single block" (2.1.1.6) function works in a similar way.

Pressing this key activates the "single block" function. The generated signal becomes active at the **end** of the (current) block being executed.
The "SBL" (Single block) display appears in the first line of the CRT display.

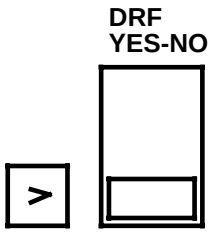
Pressing the "program start" key, causes the single block to be executed (etc. ... see above).

The following table indicates in which blocks the "decoding single block active" signal or the "single block active" signal must be present if a program is to be executed block by block.

Block	Single block	Decoding single block
Traversing blocks		
Blocks without path information		
Calculation blocks		
Switching and auxiliary function blocks		
Internally generated control blocks (CRC ...)		
Threading blocks without dry run feedrate		
Threading blocks with dry run feedrate		

= "Single block" or "decoding single block" signal is required.

- Blocks in the buffer memory which were decoded before the "decoding single block" signal was set but which have not yet been executed cannot be stopped!
- A "decoding single block" can be influenced using "OVERSTORE" (3.2.3.2).



Extend the softkey menu called under ”PROGRAM CONTROL” and press the ”DRF-HANDWHEEL-ENABLE” softkey.

Display:

DRF handwheel enable	**YES**

You have activated ”DRF” (Differential Resolver Function). It is now possible to activate the handweel for an axis which has still to be selected.

The handwheel axis assignment is defined in axial setting data (See 3.2.10).
It is only ever possible to activate only one axis with one handwheel. If a second axis is selected, the previously selected axis is automatically deactivated.

3.2.3.2 ”OVERSTORE”

Using the ”OVERSTORE” function, you can modify one or more values in the buffer memory.

”OVERSTORE” in NC STOP.
D numbers and G53-G57 can only be overstored in the RESET state.

Note:

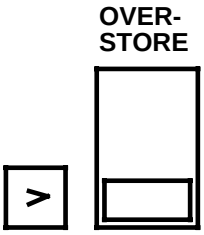
- M and S functions can now also be overstored with the extended address notation (M*=** or S*=**).
- D numbers and G53-G57 can only be overstored in the RESET state.
- Overstoring ZOF and TO in the RESET state can be used to jog to the required position with the direction keys and the actual value display.
- Overstoring of G01 and G05 is also possible in the NC STOP state. With G05, you can select oblique plunge-cut grinding and jog with the direction keys.
  Overstoring G01 (linear interpolation) can be used to deselect G05 for jogging with Cartesian coordinates again.
  In the NC program G05 is only non modal so that an interrupted automatic program is automatically resumed with G01 or G00 despite G05 having been overstored.

Sequence of operation

Prerequisite:

A program is being executed in the "AUTOMATIC" operating mode:

Press the "PROGRAM STOP" key.



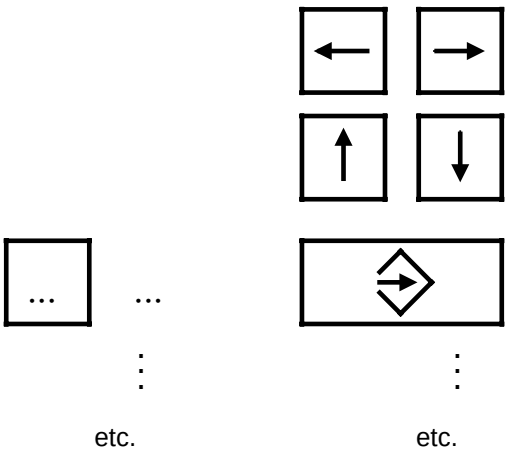
Extend the displayed softkey menu and press the "OVERSTORE" softkey .

aus 0AJ02, S. 3-80
Änderungen s. Anlage

This input form appears with the input field marked in inverse video.

Now you have the possibility to overstore the following data:

- Tool number T
- Spindle speed S
- Auxiliary function H
- Miscellaneous function M
- Preparatory function G01 or G05
  If you wish to overstore other preparatory functions or tool offset numbers D, you must first press the RESET key.



Using the cursor keys, now place the inverse marking on the appropriate input field.

Using the keyboard, enter the first corrected value and store it with the input key.

etc. etc. etc.

Press the ”program start” key when all corrections have been made.

The program is now executed with these new values until a new value in the program supersedes the overstored function, or until you enter a new value using ”OVERSTORE”.

3.2.3.3 ”BLOCK SEARCH WITH CALCULATION”

The ”BLOCK SEARCH WITH CALCULATION” function allows entry into a program at any point you wish. In the ”block search” display, you enter the program number and the block number.

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

During the block search, the same calculations are carried out as in normal program execution although the axes are not traversed!

Note:

The target position resulting from the block search with calculation is approached at a velocity set in the machine data.

Restrictions:

- In part programs with grinding-specific functions the position values for the programmed axes can be wrong after a block search with calculation. The user must therefore bring the axis positions to correct values after a block search with calculation (REPOS offset is displayed). Such functions are:
  Axis-specific feed abort (@736), several feedrates in a block, in-process measurement (@720), actual value reading (@360 - 362), reciprocation.

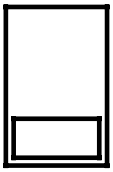
Whether the auxiliary functions are output during "block search" is set in machine data.
Depending upon the setting made on installation and start-up, the H, M, S, and T functions are output either completely or partly or not at all.

Sequence of operation

Press the "RESET" key on the machine control panel.

AUTO-MATIC

and

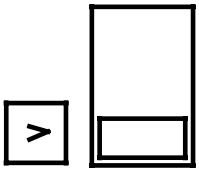


Press the operating mode selection key on the integrated machine control panel, and then the "AUTOMATIC" softkey or....

or

... rotate the operating mode selector switch on the external machine control panel to this symbol.

BLOCK SEARCH



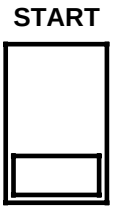
Extend the displayed softkey menu, and subsequently select the "BLOCK SEARCH" softkey function.

aus 0AJ02, S. 3-82
Änderungen s. Anlage

This screen form appears with the input field marked in inverse video.

Using the numerical keyboard, enter the programm number ”%...” and store it with the input key.

Using the numeric keyboard, enter the block number ”N...” for the search destination and store it with the input key.

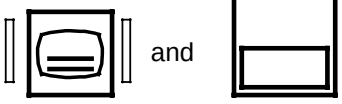


Press the ”START” softkey.

Using the ”program start” key, activate program execution.

After ”BLOCK SEARCH WITH CALCULATION”, you can traverse the difference between the actual position and the calculated command position, using the axis selector and direction keys.

Sequence of operation

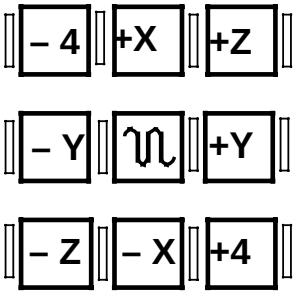


Press the operating mode selection key on the integrated machine control panel, and then the ”REPOS” softkey or...

or

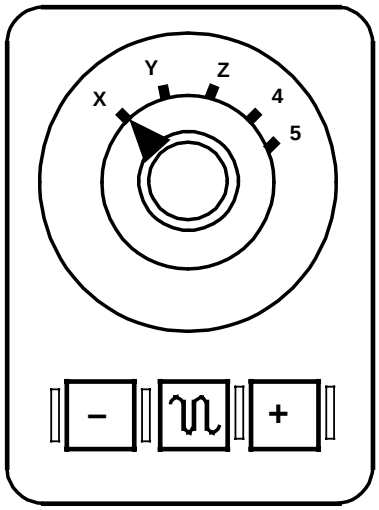
... rotate the operating mode selector switch on the external machine control panel to this symbol.

The "REPOS offset" with reference to the endpoint of the preceding block appears on the display.



Press the appropriate direction keys on the integrated machine control panel ...

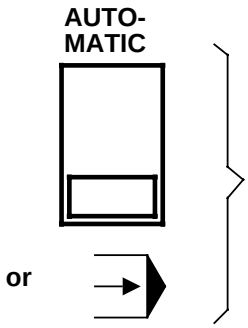
or



... select the axis to be traversed with the axis selector switch on the external machine control panel ...

... and then press one of the direction keys.

The control moves the tool using linear interpolation to the endpoint of the preceding block.



Now switch again into the ”AUTOMATIC” mode, using the softkey

or select this symbol with the operating mode selector switch.

Program processing continues.

3.2.3.4 Interrupting the program

You can interrupt program processing with ”NC-STOP”, ”feed hold” or ”spindle stop”.

Sequence of operation

”**Feed hold**”

You press the ”feed hold” key.
The feed drives are brought to a controlled standstill.

The ”feed hold” LED lights up.

Now press the ”feed start” key.
The feed is enabled again.

Note:

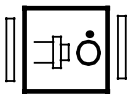
If, "feed hold" and then "feed start" are pressed, acceleration to the rapid traverse speed takes place. The retraction path is kept to while an NC block with several feedrates is being executed during retraction.

The ”feed hold” LED is also bright if there is a ”spindle stop” from the PLC and if a traverse disable is preventing start.

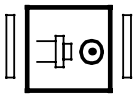
Example:

- when no servo enable has been given
- when the monitoring has been triggered
- when the feedrate override switch is at 0% **and** an enable has been given.

"**Spindle stop**"



When you press the "spindle stop" key, the spindle and feed drives are stopped.

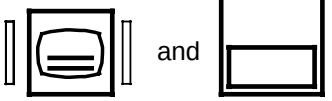


By pressing the "spindle start" key, you can enable the spindle and feed drives again.

3.2.4 Manual data input - automatic" ("MDI AUTOMATIC") mode

In this operating mode, you can execute individual operations block by block under NC control. After the blocks entered have been executed, they are deleted.

Sequence of operation

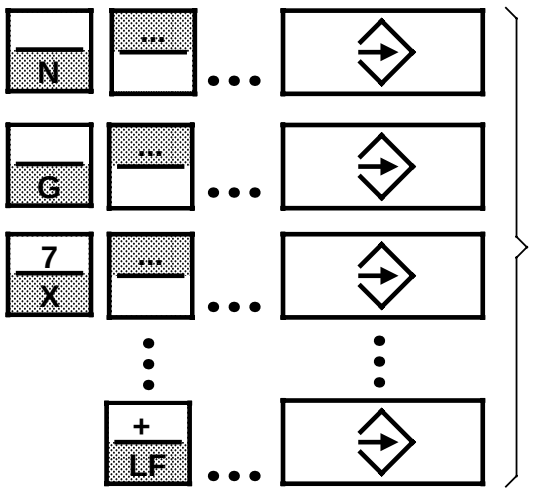


Press the operating mode selection key on the integrated machine control panel, and then the "MDI AUTOMATIC" softkey or...

or 

... rotate the operating mode selector switch on the external machine control panel to this symbol.

This screen form appears.



Using the numeric keyboard, enter a block following the guidelines in the Programming Guide and terminate each input including "LF" (end of block) using the input key.

Press the "program start" key.

"Feed hold" display must **not** be on.

The entered block is now executed by the control and subsequently deleted.

Note:

Before "program start" is pressed, you can enter several blocks (max. 256 characters).

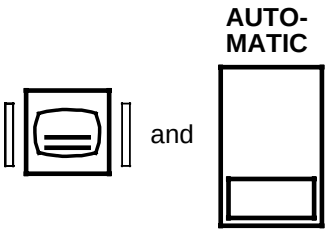
The "retentive (modal) input data" (eg. feedrate) remain. They are deleted or reset:

- by changing the operating mode
- by "RESET".

3.2.5 "TEACH IN"

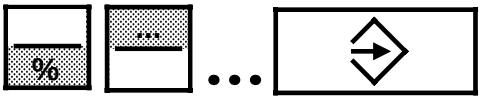
Using the "TEACH IN" function in the "AUTOMATIC"operating mode, you can generate a part program block by block. You can test the program immediately.

Bedienfolge

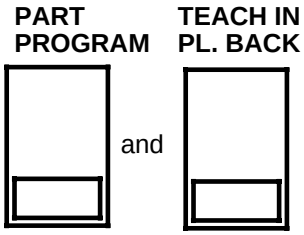


Press the operating mode selection key on the integrated machine control panel, and then the "AUTOMATIC" softkey or...

... rotate the operating mode selector switch on the external machine control panel to this symbol.

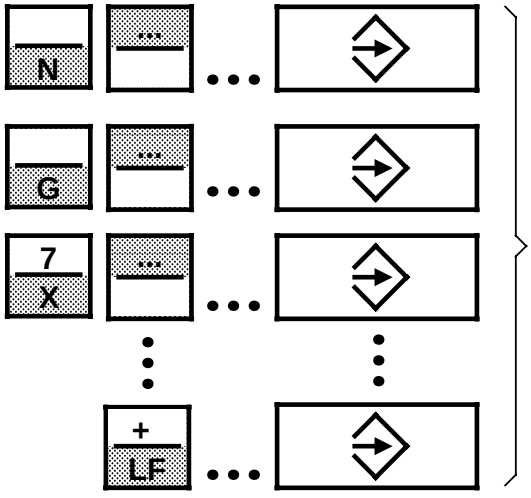


Enter the program number "% ..." (or subroutine no. "L ...") in the "AUTOMATIC" basic display.



Press the "PART PROGRAM" and "TEACH IN/PLAYBACK" softkeys, one after the other.

The selected part program No. with the input pointer below it displayed.

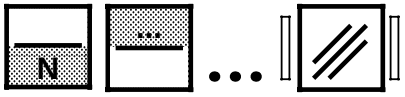


Using the numeric keyboard, enter a block (or several blocks) following the guidelines in the Programming Guide, and terminate each input - including "LF" (end of block) - with the input key.

Note :

You can also work with operator guidance, "GUIDING" (3.1.7.2).
The part program is entered directly into the part program memory.

The blocks entered can be traversed using the ”program start” key.



You can delete the blocks not required, by entering the block No. ”N ...” and pressing the delete key.
Blocks traversed correctly remain in the memory as entered. They do not have to be acknowledged.

3.2.6 ”PLAYBACK”

Using the ”PLAYBACK” function after ”TEACH IN”, you can program **linear movements.**

The position values are approached at jog traverse rate. Positions traversed are transferred into the program memory. The input resolution defines, with how many decimal places the coordinate values are transferred into the part program (up to eight decades). Decimal places can be suppressed.

The blocks entered are transferred into the program selected in the ”AUTOMATIC” basic display.

You can use ”TEACH IN” and ”PLAYBACK” alternately to create a part program. The ”TEACH IN” and ”PLAYBACK” functions are retained when changing from ”AUTOMATIC” to ”JOG” or ”INC ...” modes. They must be reselected when changing from ”JOG” or ”INC ...” to ”AUTOMATIC”!

Sequence of operation

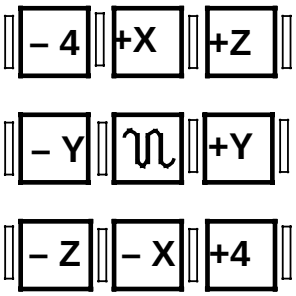
At least one complete part program block must be entered and traversed!

**1, 10, 100,
1 000, 10 000**

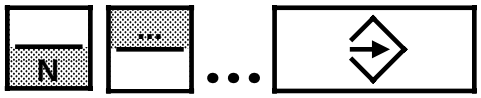
Rotate the operating mode selector switch on the external machine control panel to one of the following symbols:

JOG

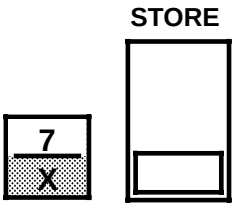
INC ...



Traverse to the desired position using the direction keys.

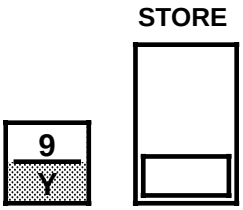


Using the keyboard, enter a block No. ”N ...”



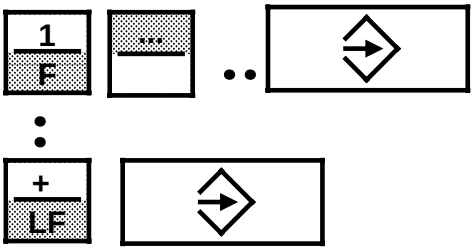
Enter the axis assignment ”**X**” using the keyboard, and then press the ”STORE” softkey.

The control transfers the X axis value under **”Actual position”** into the selected block ”N...”



Enter the axis assignment ”**Y**” using the keyboard, and then press the ”STORE” softkey.

The control transfers the Y axis value displayed under ”**Actual position**” into the selected block ”N...”.



Using the numeric keyboard, enter the required technology values, e.g. feedrate ”F...” etc., and terminate each input including ”LF” (block end) - with the input key.

Note :

You cannot work with operator guidance, ”GUIDING”.

aus 0HQ01, S. 3-99
Änderungen s. Anlage

This display shows an example of a block entered using ”PLAYBACK”.

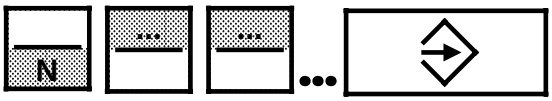
Press the ”program start” key after **each** block has been entered. This transfers the block to the program memory.

You can delete blocks not required by entering the block no. ”N...”, and pressing the delete key. Blocks traversed correctly remain in the memory as entered.

After a ”RESET” using this key, execution of the program can be repeated.

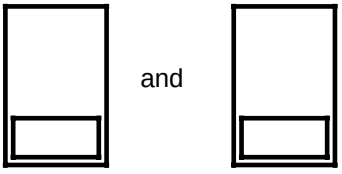
Continuing TEACH IN/PLAYBACK after RESET

Rotate the mode selector switch to the symbol for ”AUTOMATIC”.



Enter the part program number ”%...” in the ”AUTOMATIC” basic display and store it with the input key.

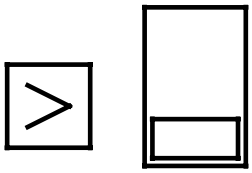
PART PROGRAM and **TEACH IN PL.BACK**



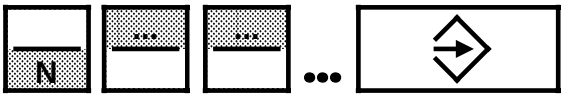
Then press the ”PART PROGRAM” softkey followed by the ”TEACH IN / PLAYBACK” softkey.

Press this key twice to return to the ”AUTOMATIC” basic display.

BLOCK SEARCH

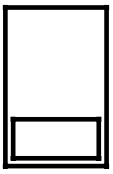


Extend the softkey menu displayed and then select the ”BLOCK SEARCH ” softkey function.



Enter the block number ”N...”of the last block in the part program and store it with the input key

START

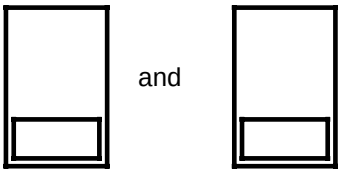


Press the ”START” softkey and ...

... start the program with the ”program start” key.

Call up the ”AUTOMATIC” basic display using these two keys

PART PROGRAM **TEACH IN PL.BACK**



and

Press the ”PART PROGRAM” and ”TEACH IN/PLAYBACK” softkeys one after the other.

Continue as described above.

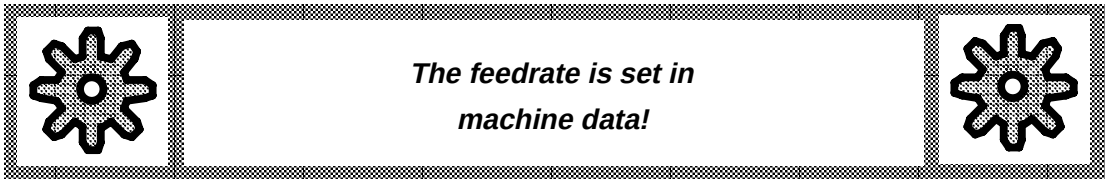
3.2.7 ”JOG” mode

In this mode, operation is NC-controlled but it is not programmed, it is manual, i. e. you **traverse** the axes with the **direction keys**!

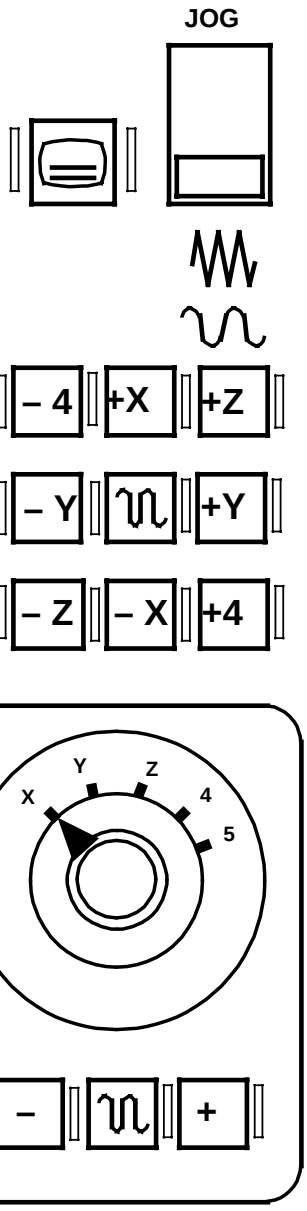
The **oblique axis** or the **oblique grinding wheel** can be traversed in a Cartesian or an oblique coordinate system. The initial setting is Cartesian motion. Overstoring G05 switches over to plunge-cut grinding. On machines with an oblique axis, only the oblique axis is moved, on machines with an oblique grinding wheel the feed axis and the reference axis are moved.

Overstoring G01 deselects G05 and thus switches to Cartesian motion.

After a program start or interruption G01 is automatically active.



Sequence of operation



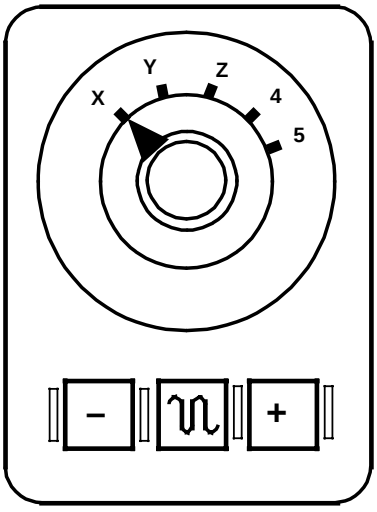
Press the operating mode selection key in the integrated machine control panel, and then the ”JOG” softkey or ...

... rotate the operating mode selector switch in the external machine control panel to this symbol.

Integrated machine control panel:

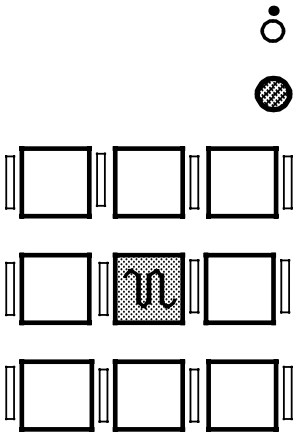
By pressing the direction keys, you cause the axes to traverse in ”jog” mode.
(You can traverse a maximum of two axes at the same time!)...

... or ...



... select the axis to be traversed with the axis selector switch on the external machine control panel, ...

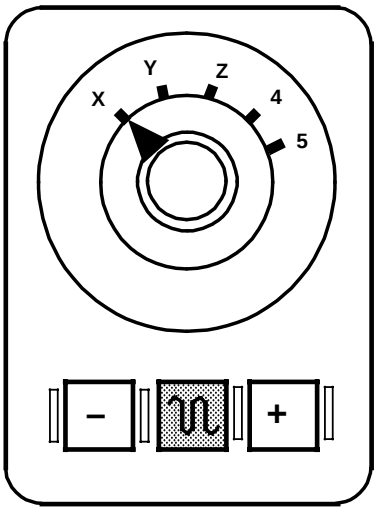
... and then press one of the direction keys.



Note:
The red ”feed hold” LED display, must **not** be bright!

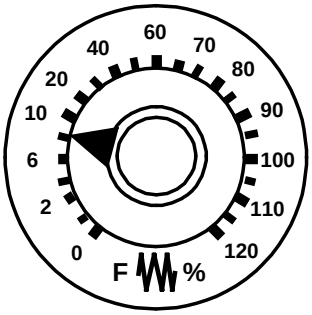
on the integrated machine control panel:

You can select ”rapid traverse” by pressing the ”rapid overlay” key simultaneously with the direction keys



on the external machine control panel:

You can press the rapid override key situated below the axis selector switch between the direction keys.



You can modify the rapid traverse rate, using the feedrate override switch.

The ”0%” setting causes feed and rapid traverse to stop.

The feedrate is set in machine data.
The feedrate can also be active for rapid traverse in the 0% to 100% range!

You can set the spindle speed, and T, H, and M functions using **”OVERSTORE”** (3.2.3.2).

3.2.8 ”Incremental” mode (”INC FEED 1 ... INC FEED 10000”)

In this mode, operation is NC-controlled but it is not programmed, it is manual; each time you press direction key, you traverse the axis concerned in the selected direction by the set increment.

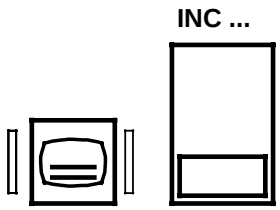
The increment can be set to the following values:

1, **10**, **100**, **1000**, **10000 increment units**.
The weighting of the increments results from the display resolution set.

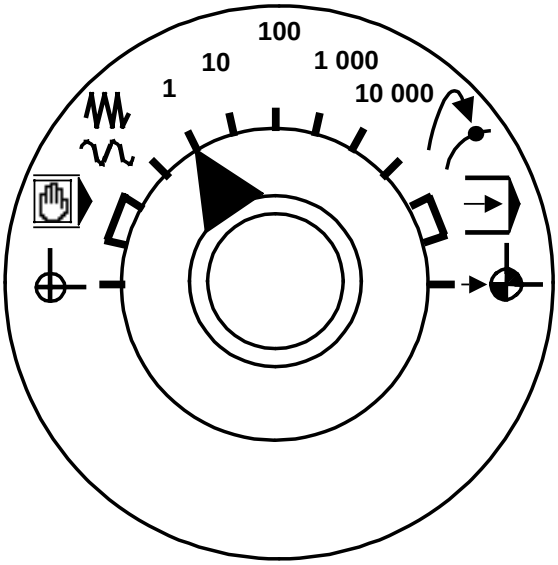
The feedrate is set in machine data.

The display resolution is set in the machine data.

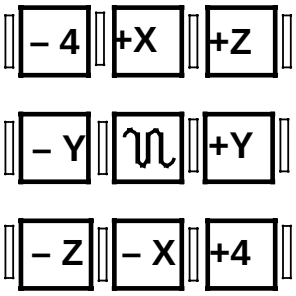
Sequence of operation



Press the operating mode selection key on the integrated machine control panel and then any of the ”**INC 1**” to **INC 10000**” softkeys or...

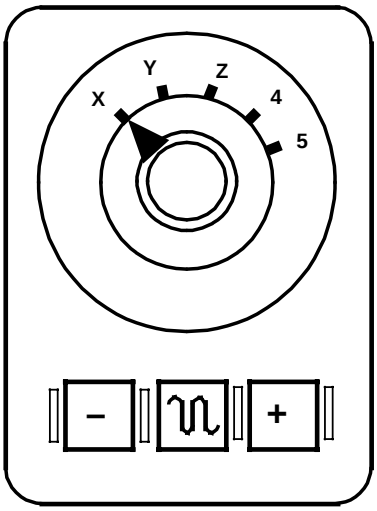


... rotate the operating mode selector switch, on the external machine control panel to the setting ”**1**”, ”**10**”, ”**100**”, ”**1000**” or ”**10000**”(”**1**” in the example) .



By pressing a direction key on the integrated machine control panel, you traverse the axis selected ...

... or ...



... select the axis to be traversed with the axis selector switch on the external machine control panel ...

... and then press one of the direction keys.

Note:

The red ”feed hold” LED display must **not** be bright!

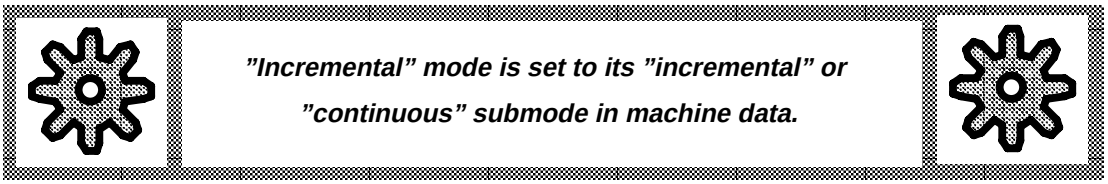
Note:

The direction keys can have two different characteristics:

- **”Incremental mode”**
- ”**Continuous mode”**

In **”incremental mode”**, when you press the direction key (howeverlong you keep it pressed), the axis only traverses by one increment (set at 1/10/100/1000/10000 increment units).

In ”continuous mode”, the axis traverses as long as you keep the direction key pressed. Traversing stops when you release the key. This also happens if the set increment has not been reached.



You can set the spindle speed, and T, H, D, G and M functions using ”**OVERSTORE**” (3.2.3.2).

3.2.9 Handwheel

Using the ”HANDWHEEL” softkey function, a command pulse from a digital-incremental encoder, connected to a handwheel, can be activated.

The increment per encoder pulse is set as for ”Incremental” (”INC ...”), however, the ”**1000**” and ”**10000**” values are **disabled**!

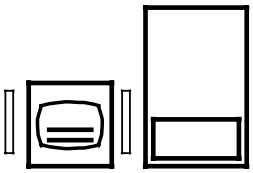
For these softkey functions, and selector switch settings, the selection is limited to ”**100**”.

The **oblique axis** or **oblique grinding wheel** can be traversed in a cartesian or in an oblique coordinate system.

For further operation see Section 3.2.7 "JOG mode" above.

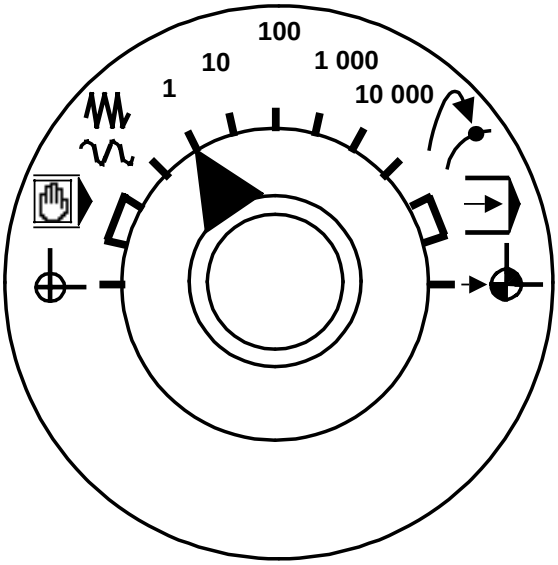
Sequence of operation

INC ...



Press the operating mode selection key on the integrated machine control panel, and then any of the "**INC 1**" to "**INC 10000**" softkeys or...

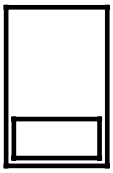
or



... rotate the operating mode selector switch on the external machine control panel to the"**1**", "**10**", "**100**", "**1000**" or "**10000**" setting.

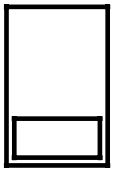
Extend the displayed softkey menu using this key.

HANDWHEEL



Select the "HANDWHEEL" softkey function.

ENABLE 1



Select the enable for the axis in the displayed softkey menu
(the 1st axis in the example).
You activate the handwheel by switching from "**NO**" to "**YES**".

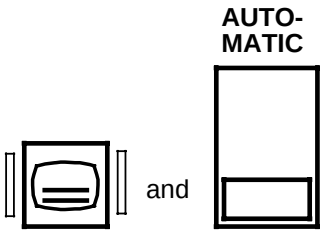
The handwheel enable ("**YES**") is displayed on the CRT.

3.2.10 ”DRF”

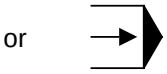
The ”DRF” (differential resolver function) softkey function makes it possible to generate an additional, incremental zero offset using the ”handwheel”.

This zero offset is **not** taken into account for the actual position display.

Sequence of operation

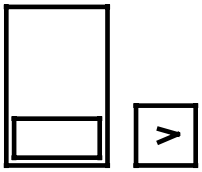


Press the operating mode selection key in the integrated machine control panel, and then the ”AUTOMATIC” softkey
or ...



... rotate the operating mode selector switch in the external machine control panel to this symbol.

Extend the displayed softkey menu with this key.

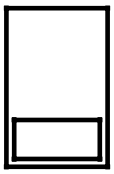


Press the ”PROGRAM CONTROL” softkey, and then extend the displayed softkey menu.

aus 0HQ01, S. 3-108
Änderungen s. Anlage

This CRT display appears.

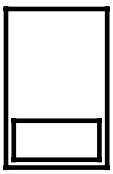
DRF
YES - NO



Press the "DRF-HANDWHEEL-ENABLE" softkey:
You activate the DRF offset by switching from "**NO**" to "**YES**".

The "DRF-handwheel" enable ("**YES**") is displayed on the CRT.

DRF

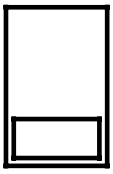


Select the "DRF" softkey function.

aus 0HQ01, S. 3-109
Änderungen s. Anlage

This display appears on the CRT.

ENABLE 1



Press the appropriate softkey in the displayed menu, to enable the axis - 1st axis in the example.

You activate the handwheel by switching from "**NO**" to "**YES**".

When you now operate the handwheel, you can effect a "DRF offset" (1st axis (X direction) in the example).

The value of the "DRF offset" is displayed on the CRT.

Do the same for the remaining axes as described above.

The handwheel pulses are weighted with the display resolution of the selected axis.

3.2.11 ”Actual value setting” mode (”PRESET”)

- In the ”PRESET” operating mode, you can offset the control zero point to any point within the machine coordinate system.

  You enter the value for the offset into the actual value memory (preset). The actual value memories for all available axes can be preset.

  This preset results in a ”PRESET offset”, which is displayed on the CRT.

- When required, a tool offset can be calculated into the ”PRESET offset”.

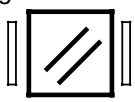
  You enter the tool offset data **before** ”actual value setting” (offset number, offset direction and type).

  The value then entered is transferred into the actual value memory taking the tool offset into account.

- **No movement** of the axes takes place with ”actual value setting”.

- The ”PRESET offset” **remains stored**:

  – after ”end of program”

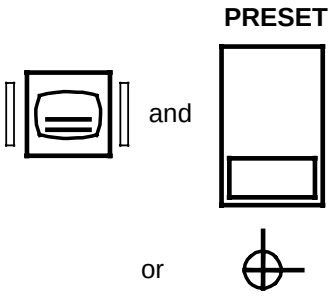
  – after ”reset”: 

Whether the ”PRESET” offset is automatically deleted on switching on the control and on traversing to the reference point, is defined in machine data.

The following value ranges must be observed for rotary axes:

Input resolution	Value range
10^{-2} degrees	0 ... 359.99
10^{-3} degrees	0 ... 359.999
10^{-4} degrees	0 ... 359.9999
10^{-5} degrees	0 ... 359.99999
10^{-6} degrees	0 ... 359.999999

Sequence of operation



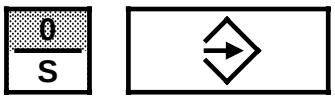
Press the mode selection key on the integrated machine control panel, and then the ”PRESET” softkey or...

... rotate the operating mode selector switch on the external machine control panel to this symbol.

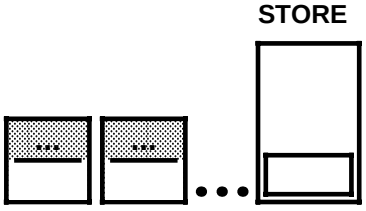
aus 0AJ02, S. 3-101
Änderungen s. Anlage

This input form with the input field marked in inverse video appears.

1. ”PRESET offset” **without** inclusion of the tool offset



Using the keyboard, enter number ”**0**” into the ”offset number” input field, and store it using the input key.

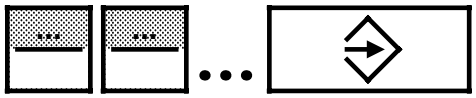


Using the keyboard, enter the **value** ”PRESET offset” (e.g. X=45) and store the value with the softkey.

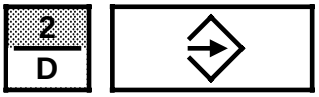
The new ”actual position” and the ”PRESET offset” for the X axis are displayed.

In the same way, you enter the values for the other axes, and then press the ”STORE” softkey for the axis concerned.

2. ”PRESET offset” **with** inclusion of the tool offset.

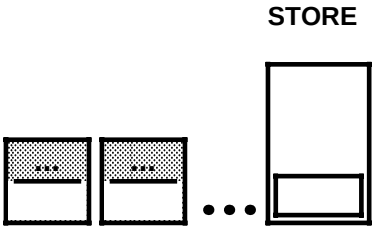


Using the keyboard, enter the number of the tool offset number D ... into the ”offset number” input field, and store it using the input key.



Using the keyboard, enter the identifier (see 3.1.3) for the X axis, into the ”Ident No.” input field - ”2” in the example. Store it using the input key.

Use the same method to enter the identification number for the other axes.

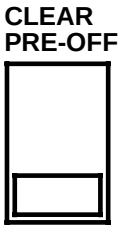


Using the keyboard, enter the value for the ”PRESET offset” (e.g. X=45).
Press the softkey to store the value for the preselected axis (”X axis” in the example).

The new ”actual position” and the ”PRESET offset” for the X axis are displayed.

The displayed value comprises:

- the entered value
- plus: the tool offset calculated into it.



You can clear the ”PRESET offset” by pressing the ”CLEAR PRESET-OFFSET” softkey.

3.2.12 ”Repositioning” mode (”REPOS”)

After a program interruption - e.g. after switching from ”AUTOMATIC” mode to ”JOG” or ”Incremental” (INC...) - you can traverse away from the contour.

”AUTOMATIC” mode is **not** aborted, i.e. the control is **not** brought to the reset state by a self-generated ”RESET”.

As described in 2.3.4, no ”RESET” is generated when you switch from ”AUTOMATIC” mode to ”JOG” or ”INC ...” or ”REPOS”. When you switch to modes other than those mentioned above, the control automatically generates a ”Reset”, which puts the control into its initial setting

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” (REPOS) mode, you can now traverse to the interruption point using the direction keys. On repositioning in G05 mode, the infeed axis must always be moved first. If this sequence is not observed in G05, alarm 3150 "Incorrect axis order" is output.

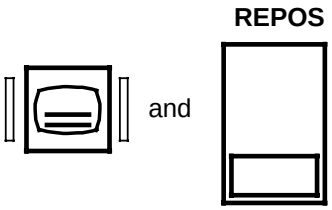
The direction key for the opposite direction is disabled, and overtravel past the start position is not possible.

After a tool change, you can only use the ”REPOS” mode with the same tool dimensions as before.
Otherwise use: – the ”block search” method (Section 3.2.3.3)
– the ”scratching” method (Section 3.2.13).

Sequence of operation

Prerequisite:

You have generated a program interruption - e.g. by pressing the ”feed hold” key - and have subsequently traversed away from the contour (see Section 3.2.7 for sequence of operation) to the ”Jog” operating mode.



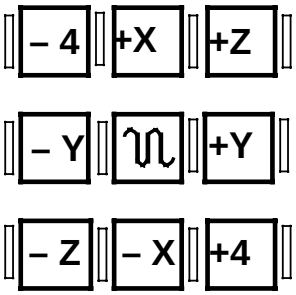
Press the mode selection key on the integrated machine control panel and then the ”REPOS” softkey or...

... rotate the operating mode selector switch on the external machine control panel to this symbol.

aus 0HQ01, S. 3-114
Änderungen s. Anlage

This display appears.

The red ”feed hold” LED display is extinguished.



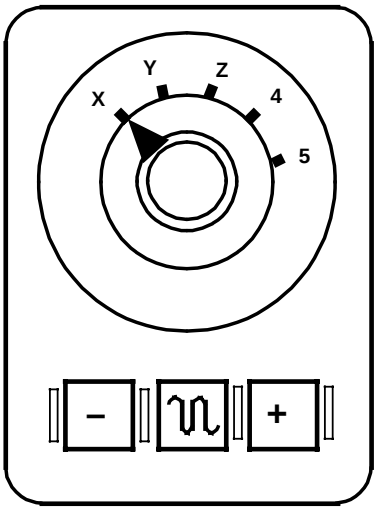
Using the direction keys on the integrated machine control panel, traverse to the point of interruption.

Note on the integrated machine control panel:

No more than two axes can be traversed simultaneously.

The feedrate override switch is active, the rapid override key is **not** active.

or...



... or ...

... select the axis to be traversed with the axis selector switch on the external machine control panel ...

... and then press one of the direction keys.

Note:

The direction keys can have two different characteristics:

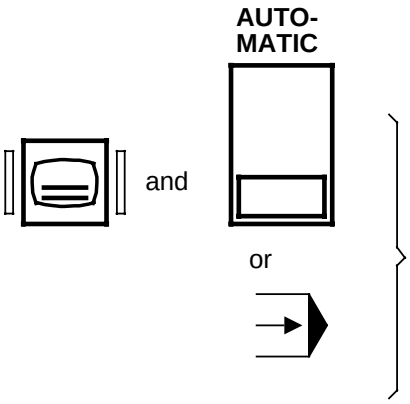
- **”Incremental mode”**
- ”**Continuous mode”.**

In **”incremental mode”**, when you press the direction key (however long you keep it pressed), the axis only traverses by one increment (set at (1, 10, 100, 1000, 10000 µm).

In ”**continuous mode”**, the axis traverses as long as you keep the direction key pressed. Traversing stops when you release the key. This also happens if the set increment has not been reached!

”Incremental” mode is set to its ”incremental” or ”continuous” submode in machine data!

You can interrupt the retraction of the axes with ”feed hold”



Using the operating mode selector key (integrated machine control panel), and softkey

or

by rotating the operating mode selector switch (external machine control panel) to this symbol, select ”AUTOMATIC” mode once the contour has been approached again.

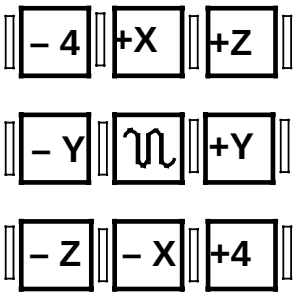
Resume program execution with ”program start”.

3.2.13 Scratching

If there is a breakdown such as a tool breakage, during program execution, you must leave the contour by changing to the ”JOG” or ”INC1...10000” operating mode, in order to change the tool.

After entering of the new tool length compensation (the grinding wheel radius remains unchanged) traverse the new tool to return to any point on the contour - within the interrupted block - (”Scratching”).

Sequence of operation



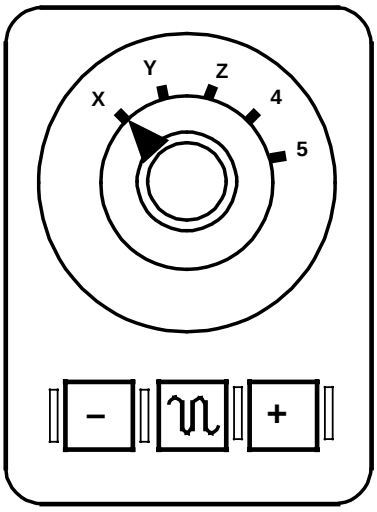
or ...

Prerequisite:

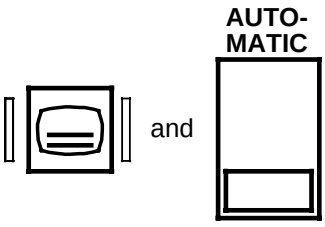
You have carried out a tool change;
the control is in ”JOG” mode:

Now traverse to any point that lies **within the interrupted** block, using the direction keys on the integrated machine control panel

... or ...



... select the axis to be traversed with the axis selector switch on the external machine control panel and then press one of the direction keys.



Using the operating mode selection key (integrated machine control panel) and softkey

or

or

by rotating the operating mode selector switch (external machine control panel) to this symbol, select ”AUTOMATIC” mode.

Resume program execution with ”program start”

In a block with circular interpolation (G02, G03) the scratching must take place within a very narrow range. This range is set in machine data (standard definition in MD ”9”). If the range is exceeded: alarm 3018!

3.2.14 Actual value display

Channel-specific actual value display

Not only are setting data, R parameters, the current block, current G functions etc. displayed but the actual value for all axes is updated in accordance with the sum of all zero offsets and the active D number in the channel displayed.

Zero offsets and tool offsets can now be programmed separately for channels 1 and 2. The actual value display refers to the channel currently selected (*).

After POWER ON RESET, the default settings for the zero offset, plane selection and tool offset number also apply to channel 2.

Note:

- Tool offsets are calculated as pure length offsets in the second channel. (No grinding wheel monitoring and tool nose radius correction).
- Overstoring D numbers and zero offsets works only in channel 1.

Zero offset display and behaviour on RESET

The actual value display calculates the values for the last active zero offset group G54-G57 in the RESET state. The corresponding tool offset is also taken into account. The actual value display does not change after program end (or RESET), if no programmable offsets were selected (G58/59).
The PRESET offset is calculated.
After NC start G54 and D0 are active.

Workpiece-related actual value system "without motion"

After selection of zero offsets (G53-G59) and selection or deselection of a tool offset (D0-D99) the actual value display is updated, even if no motion is programmed in the same block.

(*) The actual value display for channel 3 (the simulation channel) is the same as that for channel 1.

Example:

This example refers to two axes. Of course, actual value display for all axes is displayed with the function.

Default:

Setting data 5001.0 = ”1”

Zero offsets:

G54	(X, Y) =	- 54.0	additive:	G54	(X, Y) =	0.0
G55	(X, Y) =	- 0.55	additive:	G55	(X, Y) =	- 55.0
G56	(X, Y) =	0.00	additive:	G56	(X, Y) =	0.0

Tool offsets: (D1 type =1)

L1	geometric	:	P2 =	-90	L2	geometric	:	P3 =	-180
L1	wear	:	P5 =	-9	L2	wear	:	P6 =	-18
L1	basic	:	P8 =	-1	L2	basic	:	P9 =	-2

Program:			Actual value display: (X)	(Y)	Explanation:
N05	M00		54.00	54.00	After NC start G54 becomes active
N10	G55		55.55	55.55	Addition of G55 + G55
N15	G56		0.00	0.00	No ZO
N20	G58	X -500	0.00	0.00	G58 is taken over in the first block.
N25	@714		500.00	0.00	Decoding stop
N30	G55		555.55	55.55	G55 + G58
N35	G53		0.00	0.00	ZO deselection. Non modal.
N40	G58	X 0	555.55	55.55	G58 only active from the next block!
N45	G56		0.00	0.00	No ZO
N50	D1		200.00	100.00	Tool offset D1
N55	G54		254.00	154.00	Sum of G54 and D1
N60	G59	X -100	254.00	154.00	G59 only active from the next block
N65	@714		354.00	154.00	G54+G59+D1
N70	D0		154.00	54.00	Deselection of tool offset
N75	G55		155.55	55.55	G55 active
N80	M02		55.55	55.55	End of program: last zero offset group (G55) remains active. G58/59 is only active until the end of the program.

Note on tool offset:

L1 refers to the ordinate, L2 always to the abscissa of the coordinate system.

The abscissa, ordinate and applicate are assigned to the axes channel-specifically. In the NC program the assignment of the tool offset to the axes can be changed by the plane selection.

In this example, (X) is the abscissa and (Y) the ordinate.

4 Monitoring Functions

4.1 General

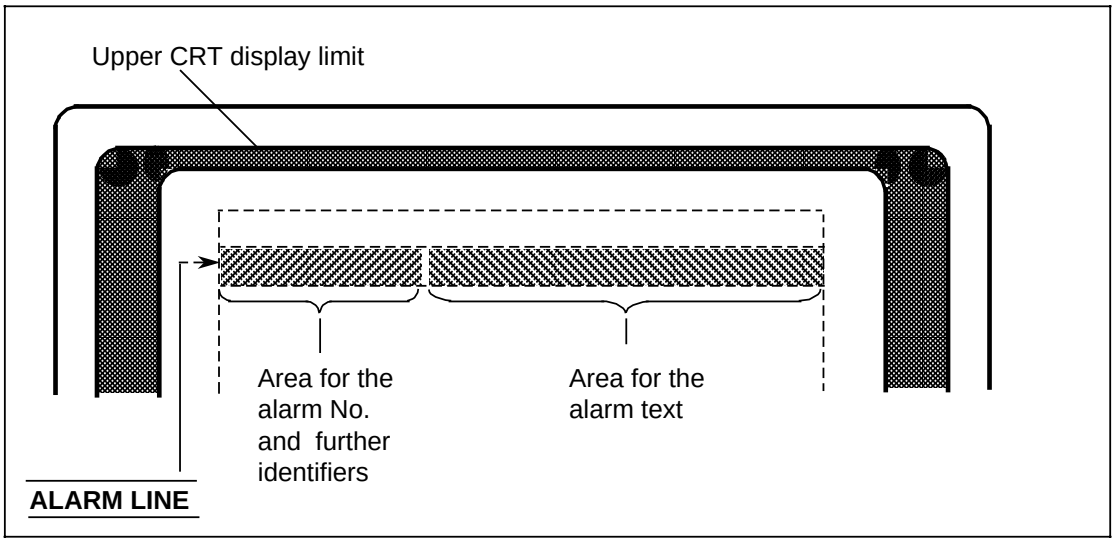
The control contains permanently active monitoring functions, which can detect faults in the NC, the interface control and the machine so early, that extensive damage to the workpiece, tool or machine are practically eliminated. If a fault occurs, first of all, machining is interrupted and the drives brought to a standstill, and the cause of the fault stored and displayed as an alarm. At the same time the PLC is informed that an NC alarm is present.

Monitoring functions exist for the following areas:

- Reading in
- Format
- Measuring loop cables
- Encoders and drives
- Contour
- Spindle speed
- Enable signals
- Voltage
- Temperature
- Microprocessors
- Serial interfaces
- Transfer between NC, COM and PLC
- Voltage level of the back-up battery
- System program memory
- User program memory.

4.2 Diagnostics display on the CRT

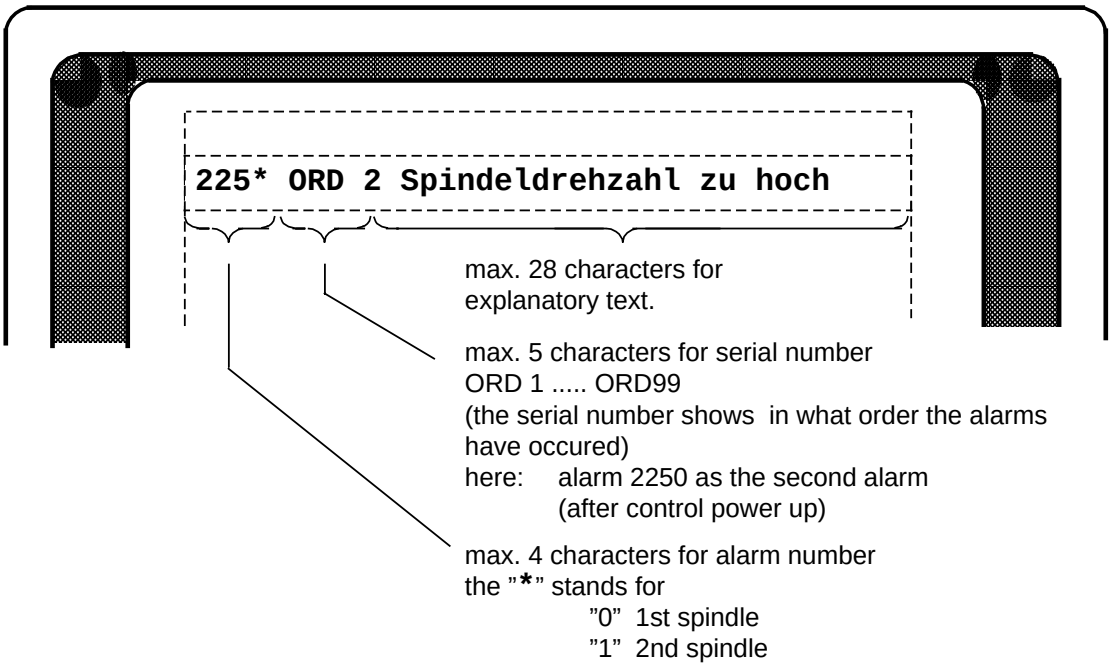
Messages from the monitoring are displayed in the ”alarm line” of the CRT display.
The ”alarm line” is the second display line from the top.



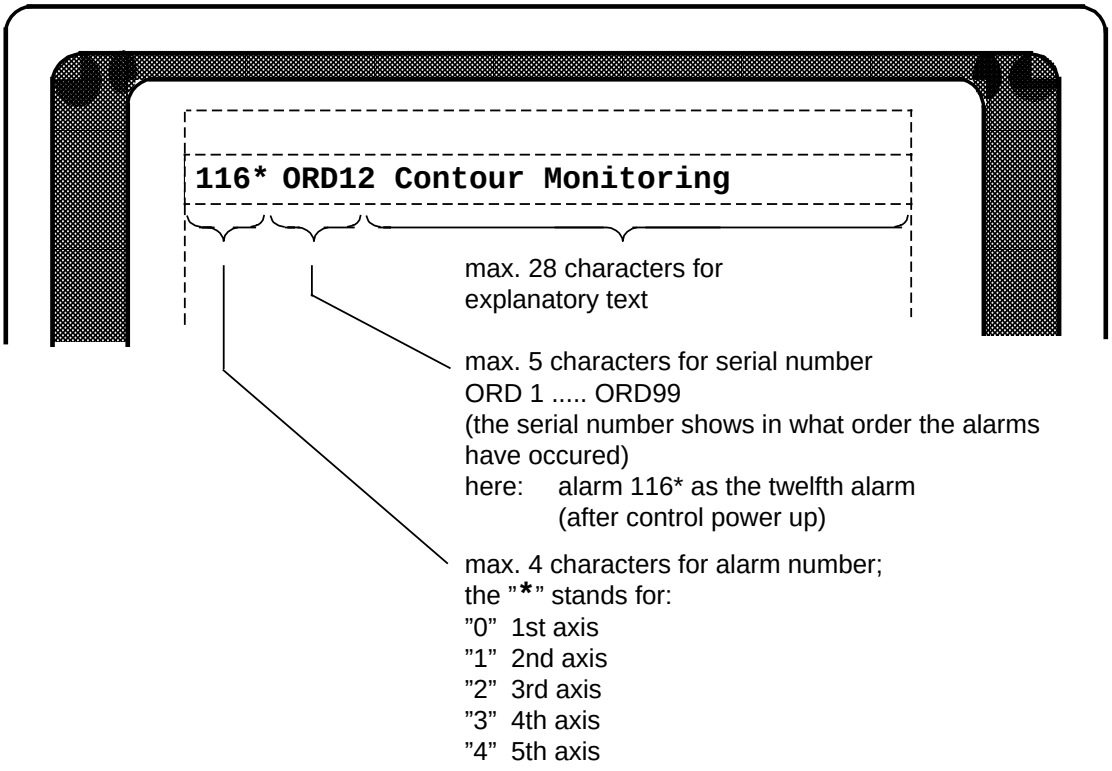
4.3 Display representation

There are 4 types of display representation:

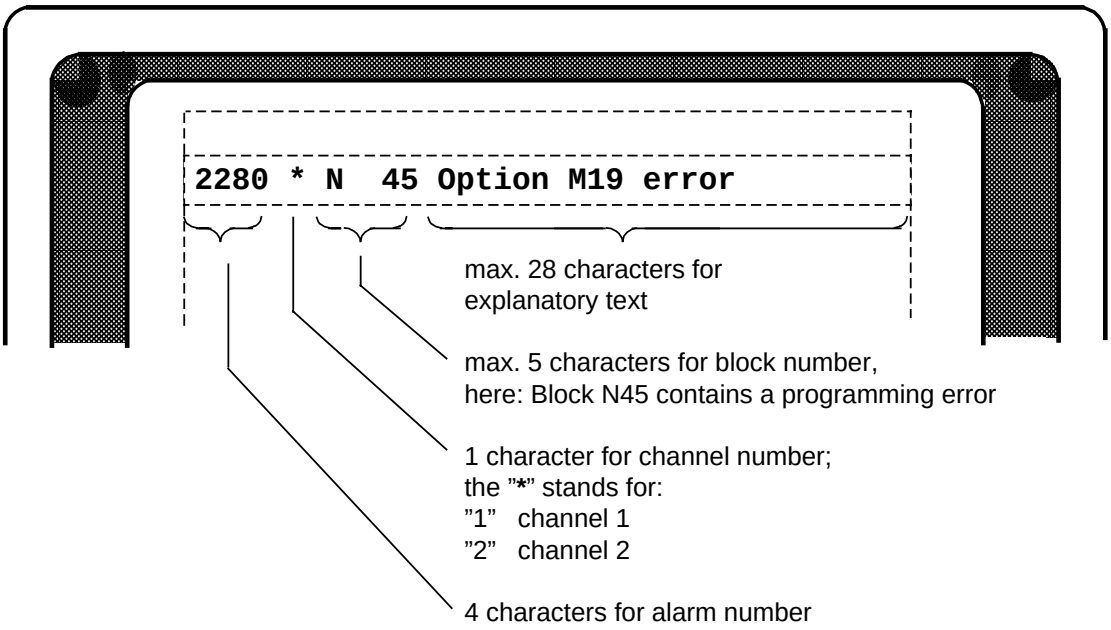
- Example of display representation type A
  Valid for alarm numbers 0 ....39 und 2000 .....2999 (partly)



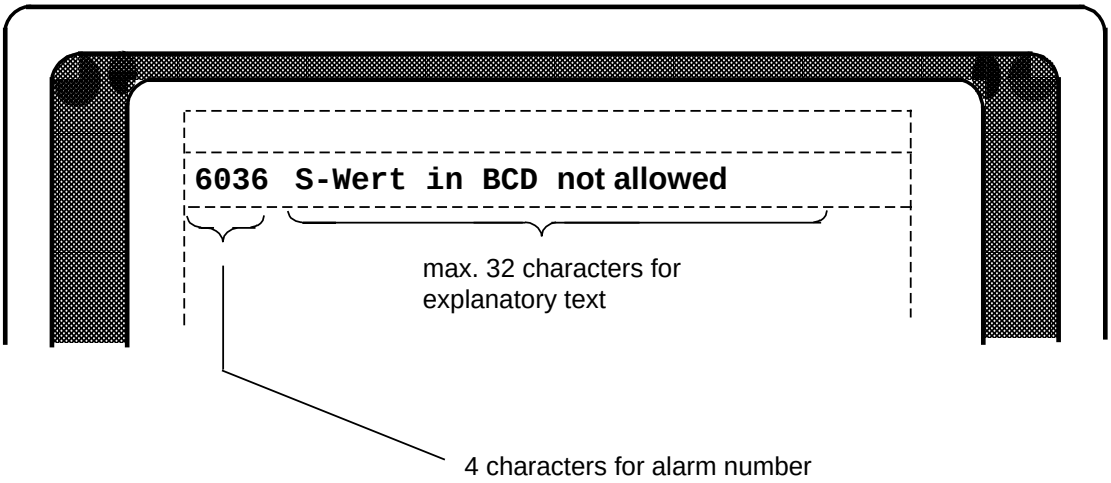
- Example of display representation type B
  Valid for alarm numbers 1000 1963



- Example of display representation type C
  Valid for alarm numbers 2000 ....2999 (partly) and 3000 ....3094 (partly)



- Example of display representation type D
  Valid for alarm numbers 6000 ....6163 (PLC error messages)
  and for alarm numbers 7000 ....7063 (PLC operational messages)



4.4 Alarm numbers and groups/delete alarms

The alarms are divided into 7 alarm groups (5 NC /and 2 PLC alarm groups)

NC alarms:
- POWER ON alarms
- RS232C (V.24) alarms
- RESET alarms/axis-specific
- RESET alarms/general
- ERASE alarms

PLC alarms:
- PLC error messages
- PLC operational messages

Alarm number	Alarm group	Alarm cleared by ...
1 .... 15 40 .... 99 132* , 226*	POWER ON alarms	Switching on the control 
1320, 1321, 1322, 1323	POWER-ON alarms/axis-specific	Switching on the control 
16 39	RS232C (V.24) alarms	1. Calling the softkey menu including the ”DATA IN-OUT function”[1)] 2. Pressing ”DATA IN-OUT softkey” 3. Pressing ”STOP softkey”
100* .... 196* [2)] außer 132*	RESET alarms/axis-specific	Pressing the RESET key 
2000 .... 2999 außer 226*	RESET alarms/general	Pressing the RESET key 
3000 3197	ERASE alarms	Pressing the acknowledge key 
6000 6163	PLC error messages	Pressing the acknowledge key 
7000 7063	PLC operational messages	The messages are reset automatically by the PLC

Tabular overview with assignment of alarm No.and clear function

---

*1) The **”DATA-IN-OUT”** function can be called in the following operating modes:*
AUTOMATIC/JOG/REFPOINT/INC1 ... INC10000/PRESET

*2) The asterisk ” ***** ” stands for : ”0” = axis 1, ”1” = axis 2, ”2” = axis 3, ”3” = axis 4, ”4”= axis 5.*

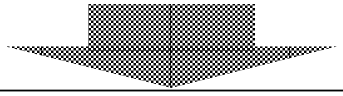
4.5 ”DIAGNOSTICS”/selection of further alarms

When the diagnostics react, the reason could be **several different faults occuring at the same time**.

Only the alarm with the **lowest alarm number** is displayed in the alarm line.

If you need an overview of any other current alarm/messages, proceed as follows:

In any of the seven operating modes, select the **DIAGNOSTICS** softkey function.



Select softkey function **NC ALARM**
or **PLC ALARM**
or **PLC MESSAGE**

4.6 Listing of the alarms / alarm description

Preliminary remark: In the following listing, it is presumed that servicing will not be carried out by the user himself.

Alarms which require a service visit are described in detail in the Installation Guide.

Alarm No. Alarm Text:	**1** ”Battery alarm power supply”
Reason: Effect: Remedy:	Backup battery voltage has dropped Backup of user memory no longer guaranteed Replace battery (See Operator's Guide)
Remarks:	Do not switch off control, data may be lost
Alarm No. Alarm Text:	**2** ”Overtemperature”
Reason: Effect: Remedy:	The temperature in the unit is too high The current NC block is completed. The next program block is not transferred for execution. Report the problem to service
Alarm No. Alarm Text:	**3** ”PLC stop”
Reason: Effect: Remedy:	PLC not operational • NC START disabled • Operation is brought to a defined standstill Report the problem to service

Alarm No. Alarm Text:	**4** ”Invalid unit system”
Remarks:	Alarm only on installation
Alarm No. Alarm Text:	**5** ”Too many input buffer parameters”
Remarks:	Alarm only on installation
Alarm No. Alarm Text:	**8** ”Wrong axis/spindle assignment”
Remarks:	Alarm only on installation
Alarm No. Alarm Text:	**9** ”Memory too small for UMS”
Remarks:	Alarm only on installation
Alarm No. Alarm Text:	**10** ”UMS error”
Remarks:	Alarm only on installation
Alarm No. Alarm Text:	**11** ”Wrong UMS Identifier”
Remarks:	Alarm only on installation
Alarm No. Alarm Text:	**12** ”PP memory wrongly formatted”
Remarks:	Alarm only on installation
Alarm No. Alarm Text:	**13** ”RAM error CPU”
Remarks:	Alarm only on installation
Alarm No. Alarm Text:	**16** ”Parity error RS232C (V.24)”
Reason: Effect: Remedy: Explanation:	The last character transferred has incorrect parity RS232C (V.24) transfer is interrupted; the last block is not stored • Check setting data 5011/5013/5019/5021 • Test external device The alarm only appears when the setting data ”with parity bit” is set.

Alarm No. Alarm Text:	**17** ”Overflow error RS232C (V.24)”
Reason: Effect: Remedy:	Transfer speed is not correct • Data transfer is interrupted • The last block is not stored • Check setting data 5011/5013/5019/5021 • Check transfer speed (baud rate)
Alarm No. Alarm Text:	**18** ”Frame error RS232C (V.24)”
Reason: Effect: Remedy	• Number of stop bits is incorrect • Baud rate is incorrect • Number of data bits is incorrect • Data transfer is interrupted • The last block is not stored • Check setting data 5011/5013/5019/5021 • Test external device
Alarm No. Alarm Text:	**19** ”I/O device not ready RS232C (V.24)”
Reason: Effect: Remedy:	DSR (Data Set Ready) signal is low No data are read in • Start external device • Do not use DSR (disconnect cable)
Alarm No. Alarm Text:	**20** ”PLC alarm memory not formatted”
Remarks:	Alarm only on installation
Alarm No. Alarm Text:	**21** ”Time monitoring RS232C (V.24)”
Reason: Remedy:	60 sec. timeout for data transfer exceeded • Check external device • Check cable • Check setting data 5017 and 5025 • Set bit No. ”0” to ”1”

Alarm No. Alarm Text:	**23** ”Character parity error RS232C (V.24)”
Reason: Effect: Remedy: Explanation:	Punched tape dirty or damaged • RS232C (V.24) transfer is interrupted • The last block is not stored Check punched tape Depending upon whether the program starts with ”%” or ”EOB”, the NC automatically identifies the ISO or EIA code upon receipt of this character and establishes the character parity. When checking the following characters, it was established that one character did **not** have the set parity.
Remarks:	Do not switch off control, data may be lost
Alarm No. Alarm Text:	**24** ”Invalid EIA character RS232C (V.24)”
Reason: Effect: Remedy: Explanation:	An EIA character with the correct parity was read in, but the character was **not** defined in EIA code • Data transfer is interrupted • The last block is invalid Check punched tape: Setting data 5026 (EIA code for ”@”) and Setting data 5027 (EIA code for ”:”) The ”=” character in EIA code is not defined, and the following data cannot be read in: %TEA1 (NC machine data) %TEA2 (PLC machine data) %PRA (R parameters) %ZOA (Zero offsets) %TOA (Tool offsets) %PCA (PLC alarm text) Main programs and subroutines with R parameter calculations
Alarm No. Alarm Text:	**26** ”Block > 120 characters RS232C (V.24)”
Reason: Effect: Remedy:	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, ...) • Data transfer is interrupted • The last block is not stored The block must be divided into two or more blocks
Alarm No. Alarm Text:	**27** ”Data input disabled RS232C (V.24)”
Reason: Effect: Remedy:	Interface signal ”cycle Inhibit” is on: NC, PLC machine data text, PLC alarm text or PLC program No data have been stored Report the problem to service

Alarm No. Alarm Text:	**28** ”Circular buffer overflow RS232C (V.24)”
Reason: Effect: Remedy:	Data transfer speed too high RS232C (V.24)transfer is interrupted, several blocks are invalid (depends on the block length) Report the problem to service
Alarm No. Alarm Text:	**29** ”Block > 254 characters RS232C (V.24)”
Reason: Effect: Remedy:	The block that has been read in has more than 254 characters. All characters (e.g. spaces) read in are counted. • RS232C (V.24) transfer is interrupted • The last block is not stored The block must be divided into two or more blocks.
Alarm No. Alarm Text:	**30** ”PP memory overflow RS232C (V.24)”
Reason: Effect: Remedy:	The maximum part program memory space is full. • Data transfer is interrupted • The last block is not stored Delete programs that are no longer required, and reorganize memory.
Alarm No. Alarm Text:	**31** ”No free PP number RS232C (V.24)”
Reason: Remedy:	The maximum number of programs defined in machine data has been reached Delete programs no longer required and reorganize memory or change machine data (service).
Alarm No. Alarm Text:	**32** ”Data format error RS232C (V.24)”
Reason: Effect: Remedy:	• The number of decades after an address is not correct • The decimal point occurs in the wrong place • The part programs or subroutines are not defined or completed correctly (check heading!) • The NC is waiting for an ”=” character, which is not defined in EIA code. Data transfer is interrupted The last block is not stored Check the data to be read in

Alarm No. Alarm Text:	**33** ”Different programs same numbers RS232C (V.24)”
Reason: Effect: Remedy: Explanation:	The stored program and the program being read in, have the same program number, but are not identical. No data are stored Either delete or ”RENAME” the old program If a new program is readin which has the same program number as one already stored, the two programs are compared. If they are different alarm 33 is displayed.
Alarm No. Alarm Text:	**34** ”Operator error RS232C (V.24)”
Reason: Effect: Remedy:	Data transfer was started on the NC, but the PLC gave a second start signal. No data are read in Stop data input and restart
Alarm No. Alarm Text:	**35** ”Reader error RS232C (V.24)”
Reason: Effect: Remedy:	Error message from Siemens Tape Reader Data transfer is interrupted The last block is not stored • Restart data transfer • If the fault reoccurs call Service.
Alarm No. Alarm Text:	**44** ”Part program memory not available”
Remarks:	Alarm only on installation
Alarm No. Alarm Text:	**48** ”PLC alarm texts from UMS illegal”
Remarks:	Alarm only on installation
Alarm No. Alarm Text:	**87** ”Illegal software limit switch”
Remarks:	Alarm only on installation
Alarm No. Alarm Text:	**100 *** ”Spindle lead grid distance invalid”
Remarks:	Alarm only on installation

** stands for: ”0”=axis 1, ”1”=axis 2, ”2”=axis 3, ”3”=axis 4*

Alarm No. Alarm Text:	**104 *** ”DAC limit”
Remarks:	Alarm only on installation
Alarm No. Alarm Text:	**108 *** ”Overflow of actual part value”
Remarks:	Alarm only on installation
Alarm No. Alarm Text:	**112 *** ”Clamping monitoring”
Reason: Effect: Remedy:	During positioning, the following error could not be eliminated faster than the time given in machine data 156. • NC START disabled • Operation will be brought to a defined stop • Follow-up mode Report the problem to service
Alarm No. Alarm Text:	**116 *** ”Contour Monitoring”
Reason: Effect: Remedy:	During acceleration or deceleration, the axis did not reach the new traverse rate inside the time fixed by the servo gain factor. • NC START disabled • Operation will be brought to a controlled stop Report the problem to service
Alarm No. Alarm Text:	**132 *** ”Control loop hardware”
Reason: Effect: Remedy:	The measuring circuit differential signals • Are not in phase • Have a short-circuit to earth • Are missing • NC START disabled • Operation is brought to a defined stop • Follow-up mode Report the problem to Service
Alarm No. Alarm Text:	**136 *** ”Measuring system dirty”
Reason: Effect: Remedy:	A measuring system with a contamination signal signals a fault to the NC. NC START The program currently running is allowed to finish. Report the problem to service

** stands for: ”0” = axis 1, ”1” = axis 2, ”2” = axis 3, ”3” = axis 4*

Alarm No. Alarm Text:	**148 * / 152 *** "+*/–* software overtravel switch"
Reason: Effect: Remedy: Explanation:	Depending upon the PLC interface signal "2nd software limit active", the 1st or 2nd software limit has been traversed to NC START disabled Traverse away from the end limit in the opposite direction The alarm is only active after reference point approach.
Alarm No. Alarm Text:	**156 *** "Set speed too high"
Reason: Effect: Remedy:	Within the control, a higher set speed was output than the value set in machine data 264*. The motor cannot follow the speed command value • NC START disabled • Operation is brought to a defined stop • Follow-up mode Report the problem to service
Alarm No. Alarm Text:	**160 *** "Drift too high"
Reason: Effect: Remedy:	The drift to be corrected by the NC is too high • NC START disabled • The green "position not yet reached" LED lights up • No traverse movement possible Perform drift compensation (service)
Alarm No. Alarm Text:	**168 *** "Servo enable traversing axis"
Reason: Effect: Remedy:	The axis-specific servo enable signal was removed by the PLC user program during traversing. • NC START disabled • Operation is brought to a defined stop • Follow-up mode Report the problem to service
Alarm No. Alarm Text:	**172 * / 176 *** "+* / –* Working area limit"
Reason: Effect: Remedy: Explanation:	The set working area limits have been reached NC START disabled • Check the program • Check the working area limits in setting data. Working area limits in "JOG" mode are set in a machine data.

** stands for: "0" = axis 1, "1" = axis 2, "2" = axis 3, "3" = axis 4*

Alarm No. Alarm Text:	**180 *** ”Axis in several channels”
Reason: Effect: Remedy:	With two programs running simultaneously in different channels, one axis was programmed in both programs (channels), causing a traversing movement for the related axis to be output for both programs. NC START disabled Check both programs
Alarm No. Alarm Text:	**184 *** ”Stop behind reference point”
Reason: Effect: Remedy:	When traversing to reference point, the axis was stopped between the reference cam and the zero mark of the measuring system • NC START disabled • Reference point not reached Approach reference point again
Alarm No. Alarm Text:	**2000** ”Emergency stop”
Reason: Effect: Remedy:	The EMERGENCY STOP signal is output from the PLC to the NC NC START disabled • Operation is brought to a defined stop • Servo enable is removed • Follow-up mode Check to see if the EMERGENCY STOP cam was traversed to, or if the EMERGENCY STOP switch was pressed
Alarm No. Alarm Text:	**2030** ”Wrong path increment”
Reason:	• Partial setpoint too large for spline. Incorrect spline coefficients K, path I or axis position • Axis offset not travelled before spline.
Alarm No. Alarm Text:	**2031** ”Eval. (weighting) factor too high (MD 388*)”
Remarks:	Alarm only on installation
Alarm No. Alarm Text:	**2032** ”Stop during threading”
Reason: Effect:	During thread cutting, the revolutional feed was stopped, which has destroyed the thread NC START disabled

** stands for : ”0” = axis 1, ”1” = axis 2, ”2” = axis 3, ”3” = axis 4*

Alarm No. Alarm Text:	**2034** ”Speed reduction area”
Reason: Remedy:	The software pre-limit was overshot, and the axis decelerated to the reduced traverse rate Check the program
Alarm No. Alarm Text:	**2035** ”Feed Limitation”
Reason: Remedy:	The programmed feedrate is higher than the contouring feedrate, based on the maximum feedrate of the axes Program a lower contouring feedrate
Alarm No. Alarm Text:	**2036** ”G35 thread lead decr. error”
Reason: Remedy:	The pitch decrease in threading, is so high that at the end of the thread, a diameter of less than, or equal to zero would result. Program either a smaller pitch decrease or a shorter thread
Alarm No. Alarm Text:	**2037** ”Programmed S value too high”
Reason: Remedy:	The programmed spindle speed ”S” is higher than ”16000” Program a smaller spindle speed (the S value is limited to ”16000” within the control).
Alarm No. Alarm Text:	**2038** ”Path feed too large”
Reason: Effect: Remedy:	Axis velocity too great because of the programmed path velocity. NC START disabled and setpoint relay drops out. Observe permissible value range.
Alarm No. Alarm Text:	**2039** ”Reference point not reached”
Reason: Effect: Remedy: Explanation:	The reference point has not been traversed to in all defined axes NC START disabled Traverse the related axes to reference point The need to traverse to reference point can be suppressed for one or more special axes with axis-specific machine data. **Attention:** Software limits are not active for these axes.

Alarm No. Alarm Text:	**2040** ”Block not in memory”
Reason: Effect:	• During block search, the required block number was not found. • During a jump in the program, the required block number could not be found in the given direction NC START disabled

Alarm No. Alarm Text:	**2041** ”Program not in memory”
Reason: Effect:	• The pre-selected program is not in the memory. • The subroutine called is not in the memory. NC START disabled
Alarm No. Alarm Text:	**2042** ”Parity error in memory”
Reason: Effect: Remedy:	One or more characters in the memory have been deleted, and can no longer be identified (these characters are output as ”?”) NC START disabled • Correct the program in EDITOR or delete the complete block and re-enter it. • When a lot of ”?” are shown, it is possible that the entire memory has been deleted; in this case, check the battery.
Alarm No. Alarm Text:	**2046** ”Block > 120 characters”
Reason: Effect: Remedy:	There is an incorrect ”LF” in the memory so that a block containing more than 120 characters exists Inhibiting of NC START Either insert ”LF”, or delete the complete block
Alarm No. Alarm Text:	**2047** ”Option not available”
Reason: Effect: Remedy:	A function was programmed that is not included in the function set of the control. NC START disabled Correct program
Alarm No. Alarm Text:	**2048** ”Circle endpoint error”
Reason: Effect: Remedy:	The programmed circle endpoint is not in the circle. The tolerance band set in machine data exceeded. NC START disabled Correct program
Alarm No. Alarm Text:	**2057** ”Option thread/revolutional feedrate not available”
Reason: Effect: Explanation:	• A thread has been programmed with G33, G34, G35, although this function has **not** been implemented in the control. • Revolutional feedrate G95 has been programmed Correct program Alarm only for SINUMERIK 810M

Alarm No. Alarm Text:	**2058** ”3D option not available”
Reason: Remedy:	In the program, three axes were programmed simultaneously. Correct program
Alarm No. Alarm Text:	**2059** ”G92 program error”
Reason: Remedy:	• Use of an invalid address character • G92 programmed with address ”P” G92 is only allowed with address ”S” (programmed spindle speed limitation)
Alarm No. Alarm Text:	**2060** ”TO, ZO program error”
Reason:	• A non-existent tool offset number was selected. • The value in the selected zero offset or tool offset is too large.
Alarm No. Alarm Text:	**2061** ”General program error”
Effect: Remedy:	NC START disabled • Check the faulty block in ”correction block”. • Select the display ”current block display” and check the blocks **following** the current block.
Alarm No. Alarm Text:	**2062** ”Feed missing/not prog.”
Reason: Remedy:	• No F value programmed • F value too small (machine data) • Feed per rev. G95 greater than 50mm/min per revolution • No feed per rev. programmed Program correct feed
Alarm No. Alarm Text:	**2063** ”Thread pitch too high”
Reason: Effect: Remedy:	A thread pitch of more than 400 mm/rev. (16 inch/rev.) has been programmed. NC START disabled Program a smaller thread pitch

Alarm No. Alarm Text:	**2064** ”Program error round (rotary) axis”
Reason: Effect: Remedy: Explanation:	If you round to either a half or full degree on a rotary axis, the control will monitor whether the programmed positions are compatible with the rounding. • NC START disabled • The programmed move in the block is **not** executed. Program the correct rotary axis position. In the JOG, INC modes the control automatically rounds to valid values. In the AUTOMATIC or MDI AUTOMATIC modes, the control only monitors the programmed positions without rounding them itself.
Alarm No. Alarm Text:	**2065** ”Position behind software overtravel”
Reason: Effect: Remedy:	The programmed block end-point lies behind the software limit switch. • NC START disabled • The programmed move is **not** executed Correct program
Alarm No. Alarm Text:	**2066** ”Thread lead increase/decrease”
Reason: Effect: Remedy:	A thread pitch increase or decrease of more than 16 mm/rev (0.6 inch/rev) has been programmed. NC START disabled Program a smaller thread lead increase/decrease
Alarm No. Alarm Text:	**2067** ”Max. speed=0”
Remarks:	Alarm only on installation
Alarm No. Alarm Text:	**2068** ”Position behind working area”
Reason: Effect: Remedy:	The programmed block end point lies, outside the working area limits, in one or more axes. • NC START disabled • The programmed motion is **not** executed • Check working area limits (plus and minus) • Alter working area limits in program with G25/G26
Alarm No. Alarm Text:	**2072** ”Incorrect input value”
Reason: Effect: Remedy:	Input for the contour definition calculation cannot be calculated. The incorrectly programmed block is not simulated or travelled. Enter correct values for the contour definition.

Alarm No. Alarm Text:	**2073** ”No intersection point”
Reason: Effect: Remedy:	Calculation of the contour path gives no intersection point with the values programmed. As for alarm 2072 As for alarm 2072
Alarm No. Alarm Text:	**2074** ”Incorrect angle value”
Reason: Effect: Remedy:	• An angle greater than or equal to 360 degrees was programmed • The angle value has no meaning for the described contour As for alarm 2072 As for alarm 2072
Alarm No. Alarm Text:	**2075** ”Incorrect radius value”
Reason: Effect: Remedy:	• Radius value too large • Radius value **not** allowed for the described contour As for alarm 2072 As for alarm 2072
Alarm No. Alarm Text:	**2076** ”Incorrect G02/G03”
Reason: Effect: Remedy:	Direction of circle for the described contour is **not** possible As for alarm 2072 As for alarm 2072
Alarm No. Alarm Text:	**2077** ”Incorrect block sequence”
Reason: Effect: Remedy:	Several blocks are needed for the calculation of the contour path: • Block sequence not correct • Insufficient information (underdefined) As for alarm 2072 As for alarm 2072
Alarm No. Alarm Text:	**2078** ”Incorrect input parameter”
Reason: Effect: Remedy:	• Programmed parameter sequence **not** allowed • Parameter sequence not complete for the described contour As for alarm 2072 As for alarm 2072

Alarm No. Alarm Text:	**2081** ”CRC/GRC not allowed”
Reason: Remedy:	With grinding wheel radius compensation selected (G41/G42), the following functions cannot be programmed: G33, G34, G35, G58, G59, G92, M19 S ... • Program G40 first • Cancel with G41/G42 D00 (GRC/CRC)
Alarm No. Alarm Text:	**2082** ”CRC plane not determinable”
Reason: Remedy:	The axes for the selected GRC plane do not exist Select correct plane with G16
Alarm No. Alarm Text:	**2087** ”Coordinate rotation not allowed”
Meaning: Remedy:	If coordinate rotation is programmed in the NC program a circular motion must be travelled immediately after changing the total angle of rotation. Check NC program.
Alarm No. Alarm Text:	**2159** ”Programming error wheel surface speed”
Reason: Remedy:	• Grinding wheel surface speed programmed for two spindles simultaneously • Spindle number and tool type do not correspond • Grinding wheel surface speed only possible for one spindle • Third decade of tool type number must correspond to spindle number • Only program grounding wheel surface speed for one spindle.
Alarm No. Alarm Text:	**2160** ”Scale factor not allowed”
Remedy:	Change scale factor
Alarm No. Alarm Text:	**2161** ”Scale change not allowed”
Remedy:	Correct scale factor modification
Alarm No. Alarm Text:	**2171** ”Approach not possible”
Remedy:	Correct approach block

Alarm No. Alarm Text:	**2172** ”Retraction not possible”
Remedy:	Correct retraction block
Alarm No. Alarm Text:	**2173** ”Wrong approach/retraction plane”
Remedy:	Correct approach/retraction plane
Alarm No. Alarm Text:	**2196** ”General reciprocating error”
Reason: Remedy: Remarks:	• The target position programmed or set in the setting data for the reciprocating axis corresponds to the current position. • G0 is active for the reciprocating axis. • Switch off reciprocation (G180) was programmed without axis designation. • The reciprocating path resulting from the programming is not allowed to be zero. • Program reciprocating axis with simultaneous feed. The block number is incorrectly displayed.
Alarm No. Alarm Text:	**2197** ”Error in coupled motion”
Reason: Effect: Remedy:	• More than one axis was declared a following axis. • Coupled motion is programmed in conjunction with G33, G34, G35. • The axis-dependent coupled motion was activated by spline interpolation G06. NC START disabled • Check machine data • Check part program
Alarm No. Alarm Text:	**225*** ”Spindle speed to high”
Scan: Explanation: Remedy:	Only when NC MD 520* bit 2 (encoder available) is set. The actual spindle speed is higher than set in machine data or setting data. • Program lower S value • NC MD 403* to 410* (max. spindle speed for 1st to 8th gear ratio) • NC MD 445* (tolerance band of maximum spindle speed) • NC MD 451* (max. spindle speed) • Check gear stage via PLC • Check G92 S... at ”v constant” • Check setting data for spindle speed limitation • Program G26 S...

Alarm No. Alarm Text:	**226*** ”Control loop spindle hardware”
Scan: Effect: Explanation: Remedy:	Cyclic • NC START disabled • Set value relay drops, set value 0 • Mode group ready 2 signal cancelled • Servo enable of the spindle is removed after the time set in MD447* has elapsed • As for Alarm 132* • As for Alarm 132*
Alarm No. Alarm Text:	**227*** ”Measuring system dirty (spindle)”
Scan: Effect: Explanation: Remedy:	Cyclic NC START disabled A measuring system with a contamination signal signals a fault to the NC. Check the measuring system
Alarm No. Alarm Text:	**228*** ”Option M19 missing”
Explanation: Remedy:	M19 S... was programmed in the part program although this function is **not** implemented in the control. • Check program • Check NC MD • Upgrade with option E42
Alarm No. Alarm Text:	**3000** ”General program error”
Reason: Remedy:	In one block of the program, a general, not precisely definable, programming error was made. Example: • An axis was programmed that is not present on the machine. • Incorrect interpolation parameters have been programmed. Check the faulty block in ”correction block”. If possible, the cursor is positioned in front of the word containing the error. The number of the block containing the error is displayed in the alarm line after the alarm number.
Alarm No. Alarm Text:	**3001** ”Geometry parameters > 5”
Reason: Remedy:	More than five geometry parameters, such as axes, interpolation parameters, radii, angles have been programmed in a block. As for alarm 3000

Alarm No. Alarm Text:	**3002** ”Polar/radius error”
Reason: Remedy:	In a block with polar/radius programming: • No angle • No radius • No coordinates for the centre point have been programmed As for alarm 3000
Alarm No. Alarm Text:	**3003** ”Invalid address”
Reason: Remedy:	An address has been programmed that is not defined in machine data. As for alarm 3000
Alarm No. Alarm Text:	**3004** ”CL800 error”
Reason: Remedy:	• @ Function not implemented • Incorrect address after the @ • Incorrect number of addresses after the @ • Value in K, R or P not permissible • Number of decades too large • No decimal point permissible • Jump destination incorrectly defined • System line (NC MD, PLC MD, TO ...) does not exist • Bit number too large • Incorrect angle data for sine or cosine • Program according to @ • Only K, R and P are valid addresses • Define jump destination forward with ”+”, backward with ”-” • Check validity of data in given addresses • Select decoded single block (DEC-SBL) and check program again
Alarm No. Alarm Text:	**3005** ”Contour definition error”
Reason: Remedy:	The coordinates in blueprint programming have been defined so that no intersection point is given. As for alarm 3000

Alarm No. Alarm Text:	**3006** ”Wrong block structure”
Reason: Remedy:	• More than 3 M functions have been programmed in one block • More than 1 S function has been programmed in one block • More than 1 T function has been programmed in one block • More than 1 H function has been programmed in one block • More than 4 auxiliary functions have been programmed in one block • More than 3 axes with G00/G01 have been programmed in one block • More than 2 axes with G02/G03 have been programmed in one block • G04 has been programmed with addresses other than ”X” or ”F” • M19 has been programmed with addresses other than ”S” • Invalid or no interpolation parameters with G02/G03 As for alarm 3000
Alarm No. Alarm Text:	**3007** ”Wrong setting data programmed”
Reason: Remedy:	• G25/G26 has been programmed • G92 has been programmed with an address other than ”S” • M19 has been programmed with an address other than ”S” As for alarm 3000
Alarm No. Alarm Text:	**3008** ”Subroutine error”
Reason: Remedy:	• Subroutine call without number of passes ”P” • M30 has been programmed as end of program • M17 missing at end of program • Four levels of nesting have been called • M17 has been programmed in a main program As for alarm 3000
Alarm No. Alarm Text:	**3009** ”Program disabled”
Explanation: Remedy:	• Subroutine not enabled for softkey start • Missing program called by PLC Set up missing program or program first program block of the subroutine @00f.
Alarm No. Alarm Text:	**3010** ”Intersection error”
Reason: Remedy:	This error can appear in conjunction with the L95 stock removal cycle, when: • A contouring program has been programmed without G00, G01, G02, G03 • @ 714 has been programmed in a contouring program • An incorrect plane has been selected in a contouring program • No intersection point has been found • More than a quarter circle has been programmed in a contouring program As for alarm 3000

Alarm No. Alarm Text:	**3011** ”Too many axes/axes twice”
Reason: Remedy:	• An axis has been programmed twice in the same block • More axes have been programmed than present on the machine As for alarm 3000
Alarm No. Alarm Text:	**3012** ”Block not in memory”
Reason: Remedy:	• The program has not been terminated with M02/M30/M17 • In a jump instruction (@100, 11y, 12y, 13y), the given block number was not found in the given direction As for alarm 3000
Alarm No. Alarm Text:	**3013** ”Simulation disabled”
Reason: Remedy:	If so set in the machine data, graphic simulation (for testing part programs) is only possible when a program is not being simultaneously run on the machine. Abort part program at an appropriate point with the RESET key.
Alarm No. Alarm Text:	**3016** ”External data input error”
Reason: Effect: Remedy:	During 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 installed Data transfer is interrupted Check PLC program or call service
Alarm No. Alarm Text:	**3017** ”Part program number occurs twice”
Reason: Effect: Remedy:	There is a program present twoce on the EPROM submodule for the cycles. NC START disabled Alarm only on installation
Alarm No. Alarm Text:	**3018** ”Distance from contour too great (NC MD9)”
Reason: Effect: Remedy:	After re-positioning to a circular contour, the distance between the axes and the contour is too great. The NC disables ”Program Start”. Re-traverse to the contour to reduce the distance.

Alarm No. Alarm Text:	**3019** ”RS232C (V.24) option not available”
Reason: Remedy:	The 2nd RS232C (V.24) interface has been started, either from the PLC or with the softkey, but the option is not installed. Transfer data via the 1st RS232C (V.24) interface. Install option C62 (2nd RS232C (V.24) interface)
Alarm No. Alarm Text:	**3020** ”Option not available”
Reason: Remedy:	A function has been programmed which is not implemented in the system. • As for alarm 3000 • Install option
Alarm No. Alarm Text:	**3021** ”CRC/GRC contour error”
Reason: Remedy:	The compensation calculation results in a traversing movement in the opposite direction to the movement programmed. Check the program.
Alarm No. Alarm Text:	**3024** ”Display description not available”
Reason: Remedy: Explanation:	A jump has been made with a programmed softkey to a display which is not present in the UMS memory or the system memory. • Check graphic display number • Check softkey function Only with additional programming on the WS 800.
Alarm No. Alarm Text:	**3025** ”Display description error”
Reason: Remedy: Explanation:	• A display with graphic elements has been programmed, without the graphics option installed in the control • The selected display has too many variables or fields • A type of display has been programmed that is not valid for the control • Check the display using the workstation • Install graphics option Only with additional programming developed in WS 800
Alarm No. Alarm Text:	**3026** ”Graphics/text too voluminous”
Reason: Remedy: Explanation:	• Configuring error in selected display • Sum of the display and text elements too large • Check the display using the workstation • Distribute the contents over two displays Only with additional programming on the WS 800

Alarm No. Alarm Text:	**3027** ”Graphics command too voluminous”
Reason: Remedy: Explanation:	The sum of the graphics commands in the display selected is too great As for alarm 3026 • This alarm triggers alarm 3026 • Only with additional programming on the WS 800
Alarm No. Alarm Text:	**3028** ”Too many field/variables”
Remarks:	Alarm only on installation
Alarm No. Alarm Text:	**3029** ”Graphics option not available”
Reason: Remedy: Explanation:	Graphic elements have been configured into the selected display, but the ”graphics” option is not installed in the control • Install ”graphics” option • Configure displays without graphic elements Only with additional programming on the WS 800
Alarm No. Alarm Text:	**3030** ”Cursor memory not available”
Reason: Remedy: Explanation:	The cursor memory programmed for the display selected is not correct (number not allowed or too large). Use the workstation to redefine the cursor memory • The function of the cursor memory is to position the cursor where it was previously when the display is called again. • Only with additional programming on the WS 800
Alarm No. Alarm Text:	**3032** ”Too many fields/variables”
Remarks:	Alarm only on installation
Alarm No. Alarm Text:	**3033** ”Display text not available”
Reason: Remedy: Explanation:	An error has occurred while linking with the workstation. Check link list, and re-link with the workstation (to clear link errors). • Only with additional programming on the WS 800 • Alarm only on installation
Alarm No. Alarm Text:	**3034** ”Text not available”
Explanation:	• Only with additional programming on the WS 800 • Alarm only on installation

Alarm No. Alarm Text:	**3040** ”Fields/variables not displayable”
Explanation: Remedy:	• Only with additional programming on the WS 800 • @410 instruction has been programmed with too many places. Check program.
Alarm No. Alarm Text:	**3041** ”Too many fields/variables”
Remarks:	Alarm only on installation
Alarm No. Alarm Text:	**3042** ”Display description error”
Reason: Remedy: Explanation:	An error has been found in the display description which cannot be exactly defined; e.g. a non-existent field has been programmed Check the display with the workstation Only with additional programming on the WS 800
Alarm No. Alarm Text:	**3043** ”Display description error”
Reason: Remedy: Explanation:	As for alarm 3042 As for alarm 3042 As for alarm 3042
Alarm No. Alarm Text:	**3046** ”Variable error”
Reason: Remedy: Explanation:	A variable has been selected that cannot be displayed in the control. Check display with workstation, re-enter variable if necessary. Only with additional programming on the WS 800
Alarm No. Alarm Text:	**3048** ”Wrong workpiece definition”
Reason: Remedy:	Minimum and maximum workpiece definition values have been interchanged in defining the workpiece Check the workpiece definition for valid values in defining the workpiece.
Alarm No. Alarm Text:	**3049** ”Wrong simulation area”
Reason: Remedy:	Incorrect or no values given for the definition of the simulation area. • Check the values for the simulation area (all planes) • A new start of simulation is possible only after RESET and pressing the ”ACKNOWLEDGE ALARM” key.

Alarm No. Alarm Text:	**3050** ”Incorrect input”
Explanation: Remedy:	• Simulation data incorrect/undefined • No reciprocation stroke on reciprocation by the PLC because of the values entered. • Check the reciprocation stroke values entered in the setting data. • Check simulation data
Alarm No. Alarm Text:	**3063** ”Data block not available”
Reason: Remedy:	A DB No. was selected in PLC STATUS which does not exist. Select correct DB. Set up DB.
Alarm No. Alarm Text:	**3083** ”Feed limit for fictitious axis”
Reason: Effect: Remarks:	On machines with an oblique axis/oblique grinding wheel, the axis velocities resulting from the programmed path feed are greater than the permissible maximum values. The path feed is reduced to travel at maximum axis velocity. The alarm can be acknowledged.
Alarm No. Alarm Text:	**3085** ”Programming error G5/G7”
Reason: Effect: Remedy:	• In the G07 block, a tool offset or zero offset was programmed. • A tool offset or zero offset was programmed before the G07 block, without the STOP DEC (@714) or the oblique axis being programmed. • G5 (G7) was programmed while grinding wheel radius compensation (G41/G42) is active. NC START disabled Correct NC program
Alarm No. Alarm Text:	**3086** ”Feed limitation”
Reason: Effect: Explanation:	An axis velocity was set which is greater than the maximum permissible velocity set in NC MD. Velocity reduced to maximum velocity. • The programmed velocity is greater than the path velocity resulting from the maximum velocity. • The ratio of follow-up path to reference path for the follow-up axis makes the feed or acceleration greater than the permissible values set in machine data. A machine data can be set to suppress the alarm.

Alarm No. Alarm Text:	**3087** ”Transformation data error”
Reason: Remarks:	• For the oblique axis/oblique grinding wheel, several or no reference axes or several or no infeed axes are defined in the machine data. • The oblique angle defined in the machine data exceeds the maximum value. The wrong machine data was entered in the block number display. Block number N5726 – reference axis MD 572* bit 6 N5727 – infeed axis MD 572* bit 7 N296 – oblique angle
Alarm No. Alarm Text:	**3150** ”Incorrect axis order”
Reason: Explanation:	This alarm only occurs in conjunction with the function oblique axis/axis with oblique grinding wheel. The sequence of the axis was not observed on repositioning. On positioning, the oblique axis/axis with oblique grinding wheel must be moved first and then the reference axis.
Alarm No. Alarm Text:	**3196** ”Dn wheel monitoring: geometry”
Reason: Effect: Remedy: Remarks:	• The minimum grinding wheel diameter entered in the tool offset n under P11 was violated. • The current wheel width stored in the tool offset n under P13 is smaller than the minimum grinding wheel width set in P12. • The tool tape entered in the tool offset n under P1 was entered. The third decade (spindle number) has an impermissible value. • The value entered in the tool offset n under P10 points to a tool offset which is outside the next display area. Program execution is stopped. Check and correct tool parameters P11, P12 and P13 in Dn. The alarm can be acknowledged once. It is thus possible to continue the current program. If the same error is detected on the next NC start the program cannot be restarted. The disable can be cancelled by overstoring D0.

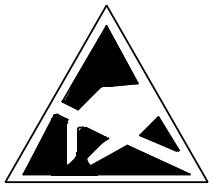
Alarm No. Alarm Text:	**3197** ”Dn wheel monitoring: rev/min”
Reason: Effect: Remedy: Remarks:	• The grinding wheel speed which the control calculates from the programmed grinding wheel surface speed and the grinding wheel diameter exceeds the maximum permissible grinding wheel speed set in the tool offset n under P14. • The grinding wheel surface speed which the control calculates form the programmed grinding wheel speed and the grinding wheel diameter exceeds the maximum permissible grinding wheel surface speed set in the tool offset n under P15. • The grinding wheel or grinding wheel surface speed programmed in the part program exceeds the value under P14 or P15. The control limits the grinding wheel speed to the smaller value from P14 compared with the grinding wheel speed value calculated resulting from the current grinding wheel diameter. Check the tool parameters P14, P15 from the part program and the spindle setpoint output. The alarm can be acknowledged.
Alarm No. Alarm Text:	**3200** ”Illegal working area limitation”
Reason: Remarks:	Values were entered in the setting data (300*, 304*) which lie outside the traverse range. The control enters the permissible maximum or minimum value in the setting data.
Alarm No. Alarm Text:	**3400** ”Check position after block search”
Explanation:	The block search was performed via grinding-specific functions. Necessary adaptation of the programmed position by external signals are not taken into account. If necessary adapt position using JOG.
Alarm No. Alarm Text:	**6000 to 6063** See explanation
Explanation:	These PLC error messages are set by the machine tool manufacturer (see machine tool manufacturer's documentation)
Alarm No.	**6100 to 6163**
Explanation	Alarm only on installation

5 Maintenance

5.1 Operating data

Humidity content to DIN 40040	F
Air pressure	860 bis 1080 hPa
Shockproof, dustproof and waterproof to DIN 40050:	
• Front of operator panel	IP 54
• Front of external machine control panel	IP 54

5.2 Handling modules



Modules/components carrying this warning symbol (yellow hand on a black triangle crossed through) are susceptible to electrostatic discharge.

Therefore, please pay attention to the following guidelines:

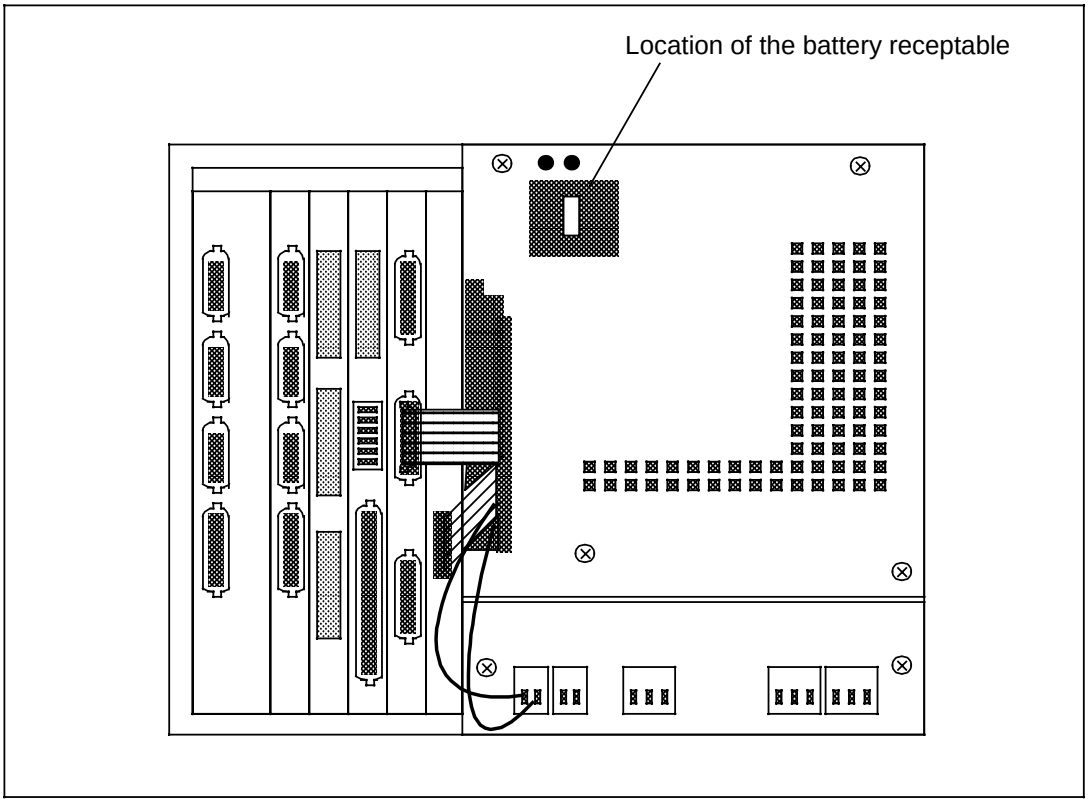
- Electronics modules should not be touched unless it is absolutely vital for the work to be done.
- Before touching a module, any static in the body must be discharged. The easiest way to do this is by touching a conductive, earthed part of the structure (e.g. bare metal parts of the cubicle, socket 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 removed or replaced 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. metallized plastic boxes, metal containers).

- If the packing is not conductive, the modules should be wrapped in conductive material. For example, conductive foam rubber or aluminium foil can be used.

5.3 Changing the battery receptacle



Schematic of the back of the SINUMERIK 810G GA3

Exchanging the battery in the battery receptacle

- The battery backs up:
  – The part program RAM memory
  – The working memory.

  The back-up time of the battery is 1 year.
- When alarm number ”1” is displayed, the battery should be replaced.
- The battery should be exchanged **within one week** of the alarm appearing!

See the Instruction Manual for the type of battery. Please order the battery from the service department of Siemens AG responsible for you.

Battery may only be replaced with the control switched on!

When changing, take care that the polarity is correct!

When disposing of batteries, certain requirements have to be met. Any queries with respect to disposal should be addressed to the local authorities.

5.4 CRT display

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

5.5 Cleaning

The front of the monitor, and the surface of the operating panel can be cleaned. For dirt that is relatively easy to remove standard household washing-up liquid or the industrial cleaner ”Special 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 a short period of time:

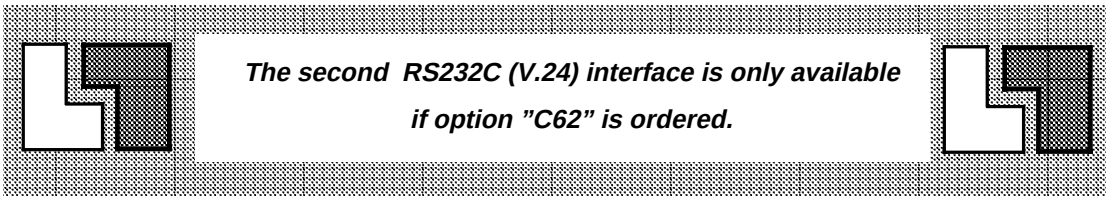
- diluted mineral acids
- bases
- alcohol
- organic hydrocarbons
- detergent solutions
- greases and oils

6 Data Interfaces

6.1 General notes

There are two universal Interfaces (RS232C (V.24)/20 mA) available for data transfer:

- **Interface 1**, on the front of the control, can be used to connect different devices.
- **Interface 2**, on the back of the control, is normally used for the connection of a particular device with fixed assignment.



The interface data (interface ”characteristics”) are set separately for each interface in the ”setting data” (SD).

6.2 Setting data for description of the interfaces

On the SINUMERIK 810G, the interfaces are described using setting data (SD).
Both interfaces ”1” und ”2” can be defined singly :

- Interface 1 : SD No. 5010 to 5017 **and** 5026 to 5029
- Interface 2 : SD No. 5018 to 5029.

The EIA Code for special characters is the same for both interfaces
(SD numbers 5026 to 5029).

The following table describes interfaces 1 and 2.

SD No. Interface 1	SD No. Interface 2	Setting data function Bit No. 7	6	5	4	3	2	1	0
5010	5018	Device coding - input							
5011	5019	Transfer format - input: Stop bit	Stop bit	Parity bits	Parity bits	Baud rate	Baud rate	Baud rate	Baud rate
5012	5020	Device coding - output							
5013	5021	Transfer format - output: Stop bit	Stop bit	Parity bits	Parity bits	Baud rate	Baud rate	Baud rate	Baud rate
5014	5022	X_{on} characters (DC start characters)							
5015	5023	X_{off} characters (DC stop characters)							
5016	5024	SSpecial Functions: Output without first X_{on} character	Progr. start with LF	End of block CR LF	Output in EIA code	Stop at "end of transmission" character	Data Set Ready (DSR) is evaluated	Output without leading and trailing tape	Input of SINUMERIK System 3 and 8 programs
5017	5025	Special Functions: Not Allocated (without function)							Time out monitor
5026	5026	EIA code for "@" character							
5027	5027	EIA code for ":" character							
5028	5028	EIA code for "end of transmission"							
5029	5029	EIA code for "=" character							

Description of the interfaces

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

Before you connect a particular device to interface 1 (or interface 2), you must set the ”setting data bits” as defined in 6.1 and 6.2

To do this, call up the relevant setting data number (SD No.) - as described in 3.1.12.1.

The following table 6.3 is to help you to make the correct setting of the ”setting data bits” for the individual devices.

Only those setting data numbers (SD Nos.) are shown in the table which are relevant for the connection of the device in question to interface 1 or 2.

Peripheral device	Ident. of the device type (CRT display)	SD No. Interface 1	SD No. SInterface 2	Required setting data bits (binary code)
SINUMERIK - Reader:T40,T50, T60, RS232C (V.24), 9600 bauds	RTS-LINE	5010 5011	5018 5019	0000 0000 1100 0111
SINUMERIK WS800, RS232C (V.24), 20 mA 9600 bauds	RTS-LINE	5010 5011 5012 5013	5018 5019 5020 5021	0000 0000 1100 0111 0000 0000 1100 0111
Siemens Programmer PG685/675/ 670/730/750 20 mA, 9600 bauds Interface PLC S5	PLC-PROG.	5010 5011	5018 5019	0000 0100 0000 0111
Siemens Programmer PG 675/685/ 730/750 RS232C (V.24), 1200 bauds Interface printer	RTS-LINE	5010 5011 5012 5013 5016 5028	5018 5019 5020 5021 5024 5028	0000 0000 1100 0111 0000 0000 1100 0111 0010 1000 0000 0011
Siemens Programmer PG615 U 20 mA, 9600 bauds	PLC-PROG.	5010 5011	5018 5019	0000 0100 0000 0111
Siemens Page Printer PT80 20 mA, RS232C (V.24), 300 bauds	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 assignments for each type of peripheral device

Peripheral device	Ident. of the device type (CRT display)	SD No. Inter-face 1	SD No. Inter-face 2	Required setting data bits (binary code)
Siemens Printer PT 88 RS232C (V. 24), 9600 bauds	RTS-LINE	5012 5013	5020 5021	0000 0000 1100 0111
SINUMERIK System 800 NC-NC link RS232C (V.24), 9600 bauds	RTS-LINE	5010 5011 5012 5013	5018 5019 5020 5021	0000 0000 1100 0111 0000 0000 1100 0111
Siemens Programming Workstation PD..., PG... RS232C (V.24), 4800 bauds	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 RS232C (V.24), 1200 bauds	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 bauds	RTS-LINE	5010 5011 5012 5013	5018 5019 5020 5021	0000 0000 1100 0000 0000 0000 1100 0000
Facit 4040, 4042 Reader/punch RS232C (V.24), 1200 bauds	RTS-LINE	5010 5011 5012 5013	5018 5019 5020 5021	0000 0000 1100 0011
Facit reader 4030 RS232C (V.24), 1200 bauds	RTS-LINE	5010 5011	5018 5019	0000 0000 1100 0100
Sommer terminal MDC-3 SNC Cassette unit RS232C (V.24), 9600 bauds	RTS-LINE	5010 5011 5012 5013	5018 5019 5020 5021	0000 0000 1100 0111 0000 0000 1100 0111

(continued)

6.4 Device connection data

SINUMERIK reader T40 and T50

Cable Order No. 6FC9 340-8S.

Device data

Transfer rate	9600 bauds
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 assignment at 27 PO1:
not allocated

Jumper assignment at 27 PO2:
Jumpers 2 and 5 closed

Jumper assignment at 27 SO2:
Jumpers 1, 2, 3, 4 open

Operating notes

To start automatically from the SINUMERIK, the reader must be ready to start, the ”reader start” LED must be lit. It is not possible to stop on a precise character.

If programs are produced externally, and several programs punched onto one tape, at least 20 blanks must be left between the programs.

When punching programs from the SINUMERIK, these blank characters are automatically generated, provided that the ”output without leader tape at beginning and end” 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 bauds
Character format	1 start bit
	8 data bits
	2 stop bits

Settings

DIP switch in unit

1	2	3	4	5	6	7	8
OFF	OFF	OFF	OFF	OFF	OFF	ON	OFF

Operating notes

Before switching the device on, ensure that the paper tape has been inserted. If there is a fault, the green LED flashes. Acknowledge by switching the device off and on.
Read-in start is controlled by the SINUMERIK.

Further operating conditions

The device stops on a precise character.

SINUMERIK WS 800

Cable Order No: 6FC9 344-1B.
(RS232C (V.24))
6FC9 344-1Q. (20 mA)

Device data

Interface: RS232C (V.24) or
20 mA current loop (TTY)
Transfer rate 9600 bauds

Character format 1 start bit
8 data bits
2 stop bits

Setting of the interface module DF 20

Interface SS2

RS232C (V.24): no hardware setting
TTY: passive
Socket X9
Jumper 2-14
Receiver
3-13
6-10 Transmitter
7-9.

Siemens Programmer PG 675, 685

Interface **printer**

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

Device data

Interface RS232C (V.24)

Transfer rate 1200 bauds

Character format 1 start bit
8 data bits
2 stop bits
1 parity bit
(even parity)

Operating conditions

The archiving of NC cycles and programs onto disk, and the transfer to and from the NC and the creation of programs is possible with the PG 675.
For data transfer, the PG 675 should be connected from the **printer** interface to an RS232C (V.24) interface on the SINUMERIK.

Siemens programmer PG 685/675/670

Interface **S5 PLC**

Cable Order No.: 6FC9 340-8G

Device data

Interface
20 mA current loop
Transfer rate: 9600 bauds
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 programmer should be directly connected to interface **1** (20 mA current loop interface) on the SINUMERIK.

Siemens programmer PG 615U 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 bauds
Character format 1 start bit
8 data bits
1 parity bit
(even parity)
2 stop bits

Operating conditions

For PLC programming the PG 615 programmer is connected via the adapter and a cable to interface **1** (20 mA current loop) on the SINUMERIK.

SIEMENS page printer PT 80

Cable Order No.: 6FC9 340-8C
(RS232C (V.24))
6FC9 340-8T (20mA)

Device data

Transfer rate:	300 baud
Character format	1 start bit 8 data bits 2 stop bits

Order No. for PT80 to SINUMERIK specification:

Type RS232C (V. 24):	L22751-A80-D442 (Interface module STT104)
Type 20mA:	L22751-A80-D441 (Interface module STT104+LAT101)
	Additional cable for terminal connection: 6FC9340-4KA

An NC-controlled read operation (start/ stop) is possible on a device with a 20 mA interface.

SIEMENS printer PT 88

Cable Order No.: 6FC9 340-8D.

Device data

Interface adaptation SAP-S2
(RS232C (V.24))

Setting of the operating mode switches

Switch S1

1	2	3	4	5	6
ON	ON	ON	OFF	OFF	OFF

(with this setting, the printer has a transfer rate of 9600 bauds)

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)

7 Interfacing to the Machine

7.1 General notes

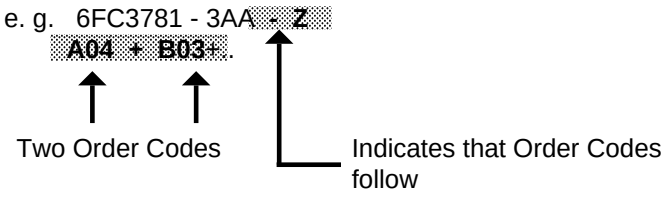
How SINUMERIK 810G GA3 is operated depends on the setting of the machine data, and the options.

In the basic version of the SINUMERIK 810G GA3, the standard machine data are set in the works. The machine data can be modified when interfacing to the machine and when options are installed.

7.2 Ordering data options

The Order Number for the basic versions of the SINUMERIK 810G GA3 control is
810G 6FC3 781-3AA

If the control is equipped with options, the Order Number is **extended**.
It then has the following format:



Tick the boxes in the column

Options supplied with control?

in the following table if the order number includes the order code in question.

Identification No. of the NC control:
(serial No., ident., inventory No. etc.)

Function of the option	Order code	Option supplied with control?	Comments
3rd axis	A03		
4th axis	A04		
5th axis	A05		
Tape reader type T61	B01		
Tape reader type T40	B02		
Tape reader type T50	B03		
3-D interpolation, Helical interpolation	B61		
Cylindrical grinding cycles	B71		
Program memory expansion	C47		
2nd RS232C (V.24) interface	C62		
Oriented spindle stop	E42		
Constant grinding wheel surface velocity	E43		
Oblique plunge-cut grinding	E85		
Output of reversal signals	E86		
Analog spindle speed spindle 1	F05		
Analog spindle speed spindle 2	F06		
External data input	F72		
Leadscrew error compensation	H56		

Identification No. of the NC control:
(serial No., ident., inventory No. etc.)

Function of the option	Order code	Option supplied with control?	Comments
Graphic simulation	J16		
Display texts in:			
English	---		
German	J22		
French	J23		
Italian	J24		
Spanish	J25		
Dutch	J26		
Russian	J27		
Swedish	J28		
Integrated machine control panel	J81		
Integrated customer machine control panel	J82		
External machine control panel	J85		
2nd spindle submodule	J90		
Measuring-circuit module with 3 control loops without EXE, not upgradable with EXEs	K20		
Measuring-circuit module with 3 control loops without EXE, not upgradable with EXEs	K21		
Measuring-circuit with 3 control loops with 1 EXE, 5-fold/10-fold	K23		
Measuring-circuit with 3 control loops with 2 EXEs, 5-fold/10-fold	K24		

Identification No. of the NC control:
(serial No., ident., inventory No. etc.)

Function of the option	Order code	Option supplied with control?	Comments
Measuring-circuit with 3 control loops with 3 EXEs, 5-fold/10-fold	K25		
I/O submodule, basic module with mounting plate 64I/32Q	M01		
I/O submodule, basic module without mounting plate 64I/32Q	M02		
I/O submodule, basic module without mounting plate 64I	M03		
I/O submodule, basic module with mounting plate 64I	M04		
Interface submodule for electronic handwheels	M10		
Interface submodule for handwheel with mounting plate	M11		
Round cable 1 m (for I/O submodules)	M49		
Round cable 5 m (for I/O submodules)	M50		
Ribbon cable for connecting: 1 submodule	M51		
Ribbon cable for connecting: 2 submodules	M52		
Ribbon cable for connecting: 3 submodules	M53		
Ribbon cable for connecting: 4 submodules	M54		
Ribbon cable for connecting: 5 submodules (5th submodule handwheel)	M55		

Identification No. of the NC control:
(serial No., ident., inventory No. etc.)

Function of the option	Order code	Option supplied with control?	Comments
Integrated PLC auxiliary axes	N06		
UMS RAM 128 Kbytes	N23		
256 Kbytes	N24		
Configurability UMS EPROM 128 Kbytes	N31		
Configurability UMS EPROM 256 Kbytes	N32		
INPUT/OUTPUT module for PLC exp. device 32 inputs type K	N60		
INPUT/OUTPUT module for PLC exp. device 32 inputs type U	N61		
INPUT/OUTPUT module for PLC exp. device 32 inputs 0.5 A type K	N65		
INPUT/OUTPUT module for PLC exp. device 32 inputs 0.5 type U	N66		
INPUT/OUTPUT module for PLC exp. device 16 inputs 2A type U	N68		
INPUT/OUTPUT module for PLC exp. device 16 inputs 1A type K	N70		
INPUT/OUTPUT module for PLC exp. device 64 inputs type SINUMERIK	N71		
INPUT/OUTPUT module for PLC exp. device 32 inputs 0.5 A type SINUMERIK	N72		
INPUT/OUTPUT module for PLC exp. device 32 inputs 2 A type SINUMERIK	N73		

Identification No. of the NC control:
(serial No., ident., inventory No. etc.)

Function of the option	Order code	Option supplied with control?	Comments
Link to SIMATIC S5 (host PLC)	**N75**		
MPC link module (in the maxi EU)	**N96**		
PLC mini EU	**P06**		
Distributed linking of I/O sub-modules (MPC header submodule)	**P07**		
PLC expansion device (maxi EU)	**P08**		

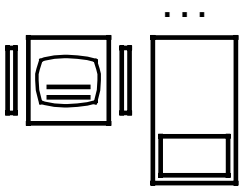
7.3 Machine data

7.3.1 General machine data

You must only check the machine data displayed,
you must not modify them!

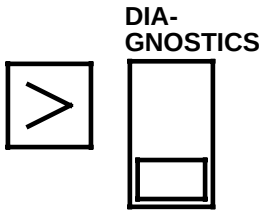
To call up the CRT display, proceed as follows ...

Sequence of operation: calling general machine data



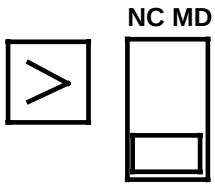
Press the mode selection key on the integrated machine control panel, and then **any operating mode** softkey.

or select **any operating mode** using the mode selector switch (external machine control panel).



Press the key to extend the softkey menu, and then the ”DIAGNOSTICS” softkey.

In this section you will find the operating steps for displaying machine data on the screen. The meaning of the machine data is described in the service documentation "Installation instructions for the SINUMERIK 810G GA3/820G GA3".

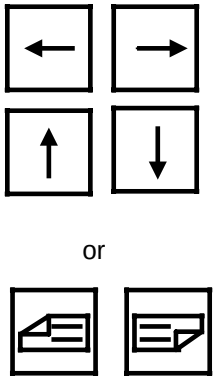


Press the key to extend the sofkey menu, and then the ”NC MACHINE DATA” softkey.

The ”general machine data” display appears.

Using the keyboard, now enter the number of the machine data and press the search key.

The machine data searched for, and the value stored in the control are displayed in inverse video.



You can also use the cursor keys

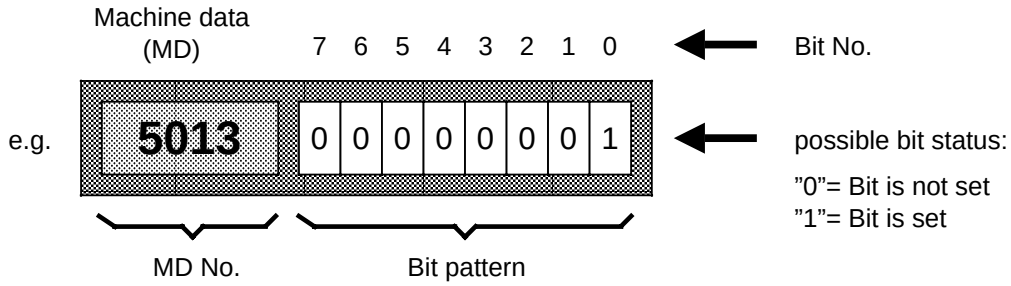
or

the ”page backwards/forwards” keys to call further machine data.

Consult the machine tool manufacturer for the settings and meanings of the maching data.

7.3.2 Machine data bits

The machine data bits are structured as follows:



You must only check the bit pattern displayed.

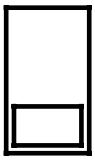
You must not modify it!

To call the CRT display, proceed as follows ...

Sequence of operation: calling machine data bits

The sequence of operation is as in Section 7.3.1 for: "general machine data", as far as pressing the "NC MACHINE DATA" softkey and then ... .

MACHINE BITS

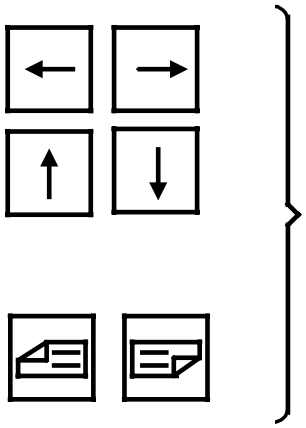


... press the "MACHINE DATA BITS" softkey.

The "machine data bits" display appears.

Using the numeric keyboard, now enter the number of the machine data and press the search key.

The machine data searched for and the bit pattern stored in the control are displayed in inverse video.



You can also use the cursor keys

or

the ”page up/down ” keys to call further machine data.

Consult the maching tool manufacturer for the settings and meanings of the machine data.

8 Appendix

8.1 List of abbreviations

Abbreviation	Meaning
BCD	Binary coded decimal
CH1	Channel 1, main channel
CH2	Channel 2, auxiliary channel
CH3	Channel 3, simulation channel
CLF	Clear file
CPU	Central processing unit
CRC	Cutter radius compensation
DAC	Digital/analog convertor
DIO	Data input/output
DRF	Differential resolver function
EIA code	Special tape code, number of holes per character is always odd
EOB	Label for end of block for data in EIA code
EOR	Label for the program number (when EIA code is used)
EPROM	Program memory with fixed program (erasable programmable read only memory)
GRC	Grinding wheel radius compensation
INC	”Incremental” mode
ISO code	Special tape code, number of holes per character is always even
LED	Light emitting diode
LF	Label for end of block for data in ISO code (line feed)
MD	Machine data
MDI	Manual data input
MIB	Machine input buffer
MPF	NC part program (main program file)
NC	Numerical control
PCA	PLC alarm text
PCP	PLC program in machine code (programmable control program)
PLC	Programmable logic control

Abbreviation	Meaning
Pr Stop	Display of a programmed interruption of the program sequence
RAM	Program memory which can be written into and read from (random access memory)
RPA	R parameter numbers with value assignment (R parameter active)
SEA	Addresses with value assignment (setting data active)
SPF	Subprogram (subroutine) file
SW limit switch	Software limit switch
TEA1	NC machine data (testing data active 1)
TEA2	PLC machine data (testing data active 2)
TO	Tool offset
TOA	Tool offset (tool offset active)
UMS	User memory submodule
ZO	Zero offset
ZOA	Zero offset (zero offset active)

8.2 Lists of terms used

Term	Section
Acknowledge alarm	2.1.1.5
ACTUAL (CURRENT) BLOCK	2.5
ACTUAL (CURRENT) VALUES	2.5
Actual position	2.1.1.5
Address/numeric keys	2.1.1.3
Alarm messages	2.6
Automatic tool offset	3.1.4.2
Automatic zero offset	3.1.5.1
AUTOM. TO	2.5
AUTOM. ZO	2.5
Basic dimension	3.1.4
BLOCK END	2.5
BLOCK NUMBER	2.5
BLOCK SEARCH	2.5
Block search	3.2.3.3
Back-up batteries	5.3
Channel switchover	2.1.1.5
CIRCLE	2.5
Cleaning	5.5
CLEAR PRE-OFF	2.5
CONTOUR	2.5
Contour definition	3.1.8
Control keys	2.1.1.5
COPY	2.5
CORR. BLOCK	2.5

Term	Section
CRT display	2.1.1.1
Cursor	2.1.1.5
CYCLES	2.5
DATA IN-OUT	2.5
DATA IN START	2.5
Data input/output	3.1.12
DATA OUTPUT	2.5
DEC.SBL YES/NO	2.5
DELETE	2.5
Delete input	2.1.1.4
Delete word/block	2.1.1.4
Device interface data	6.4
Device setting data	6.2
DIAGNOSTICS	2.5
Differential resolver function (DRF)	3.2.10
Direction keys	2.1.1.6/2.1.2.1
DIRECTORY	2.5
Display field	2.1.1.2
DRF	2.5
DRF YES/NO	2.5
DRY YES/NO	2.5
EDIT	2.5
Edit and input keys	2.1.1.4
Edit word	2.1.1.7
EMERGENCY STOP	2.5
External machine control panel	2.1.2/2.1.2.1
FEED	2.5
Feed	2.1.1.6
Feed/jog	2.3.2

Term	Section
Feed/rapid override	2.1.2.1
Feed hold/feed start	2.1.1.6/2.1.2.1
G FUNCTIONS	2.5
Guidance with input	3.1.7.2
GUIDING	2.5
HANDWHEEL	2.5
Handwheel	3.2.9
HANDWHEEL-AXIS	2.5
Ident. No.	3.1.4
”Incremental” mode (INC...)	3.2.6
Incremental Jog 1... 10 000	2.3.2
Input character/word	2.1.1.4
Installation/start-up	2.1.1.5
Integrated machine control panel	2.1.1.6
Interfaces	6
JOG mode	3.2.7
Keyswitch	2.1.2.1
LED: ”Feed hold”	2.1.1.2
LED: ”Key assignment”	2.1.1.2
LED: ”Monitoring active”	2.1.1.2
LED: ”Position not yet reached”	2.1.1.2
LED: ”Program running”	2.1.1.2
LINE	2.5
MACH. CYCLE	2.5
MACH. DATA	2.5
Machining data	7.3
MAIN PR. START	2.5
Maintenance	5
Manual data input automatic	2.3.2
MDI automatic	3.2.4

Term	Section
Menu tree	2.3.5
Menu trees for the operating modes	2.4
Modify word	2.1.1.4
Monitoring	4
MOVE	2.5
NC ALARM	2.5
NC MD	2.5
Operating data	5.1
Operating modes	2.3
Operating modes - Overview	2.3.2
Operating mode selection	2.1.2.1/2.3.3
Operator interface	2.1.1
Options	2.1/7.2
OPT. STP. YES-NO	2.5
OVERR. YES/NO	2.5
OVERSTORE	2.5
Overstore	3.2.3.2
Paging	2.1.1.5
PART PROGRAM	2.5
PLAYBACK	2.5
Playback	3.2.6
PLC ALARM	2.5
PLC MD	2.5
PLC MESSAGE	2.5
PLC STATUS	2.5
PROGR. CONTROL	2.5
Program control	3.2.3.1
Program edit	3.1.9
PROGR. END	2.5

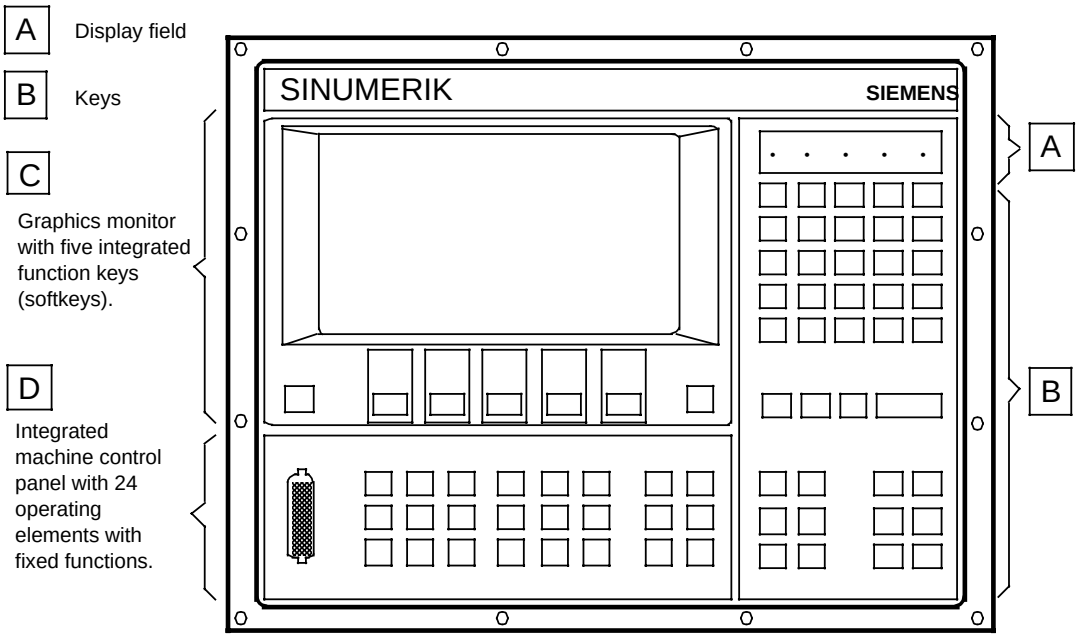
Term	Section
PROG. HANDL.	2.5
Program input	3.1.7
Program interruption	3.2.3.4
Program management	3.1.10
Program memory	3.1.10.8
Program stop/program start	2.1.1.6/2.1.2.1
RS232C (V.24) interface	6.1
Rapid traverse	2.1.1.6
Reference point approach	2.3.2/3.1.2
RENAME	2.5
REORG	2.5
Repositioning	2.3.2
Repositioning (REPOS)	2.3.22
RESET	2.5
Reset	2.1.1.6/2.1.2.1
Reset on change of mode	2.3.4
R PARAMETER	2.5
R Parameters	3.1.6
Scratching	3.2.13
Search address, block No., word or data	2.1.1.5
Sequence of operation	3
Sequences of operation, machining	3.2
Sequences of operation, preparation	3.1
SERVICE AXES	2.5
Settable zero offset	3.1.5
Setting actual value	2.3.2
Setting actual value (Preset)	3.2.11
SETTING DATA	2.5
Setting data	3.1.5/3.1.6/6.2
Setting data R parameter	3.1.6
Setting data zero offset	3.1.5

Term	Section
SIMULATION	2.5
Simulation	3.1.11
Single block	2.1.1.6/2.1.2.1
SINGLE BLOCK	2.5
SKIP YES/NO	2.5
Softkey functions	2.5
Softkeys	2.1.1.1
SPECIAL BLOCK	2.5
SPECIAL FUNCT.	2.5
SPINDLE	2.5
Setting data spindle	3.1.6
Spindle speed	2.1.1.6
Spindle speed override	2.1.2.1
Spindle stop/spindle start	2.1.1.6/2.1.2.1
START	2.5
STOP	2.5
STORE	2.5
STORE AXES	2.5
SUBROUTINE	2.5
SUBROUT. START	2.5
Switching off	2.2.2
Switching on	2.1.2.1/2.2.1
Teach in	3.2.5
TEACH IN	2.5
THREAD	2.5
TOOL	2.5
Tools	3.1.3
Tool offsets	3.1.4
Tool number	3.1.4.1

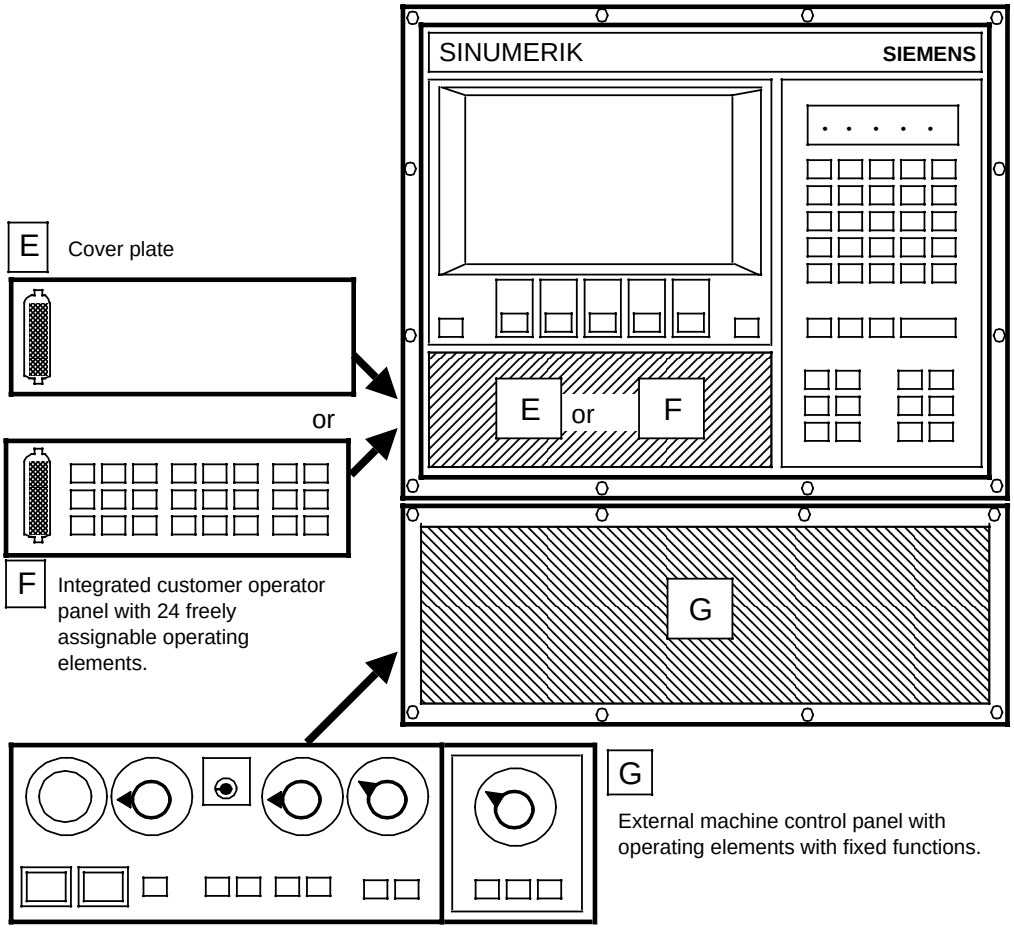
Term	Section
TO	2.5
Universal interface	2.1.1.6
WORKPIECE	2.5
WORKPIECE AREA	2.5
ZERO OFFSET	2.5
Zero offset	3.1.5

8.3 SINUMERIK 810G GA3 operator interface complete view

SINUMERIK 810G GA3 with integrated machine control panel



SINUMERIK 810G GA3 with external machine control panel



Siemens AG

AUT V250
P.O. Box 48 48
W-8500 Nuremberg 1
Federal Republic of Germany

Suggestions

Corrections

For Publication/Manual:

SINUMERIK 810G
Software Version 3
Operator's Guide

User Documentation

Order No.: 6ZB5 410-0HQ02-0BA0
Edition: May 1992

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 Aktiengesellschaft

Bestell-Nr. 6ZB5 410-0HQ02-0BA0
Printed in the Fed. Rep. of Germany
251/332008 BA 11920.2

Progress
in Automation.
Siemens