SINUMERIK System 3
Basic Version 4C

Operator's Guide

SINUMERIK

Edition
08.88

SINUMERIK System 3
Basic Version 4C

Operator's Guide

Edition 08.88

Subject to change without prior notice

Functions extending beyond the scope of this Description may be capable of operating on the controller. However, we accept no responsibility for such functions for new equipment or equipment which has been serviced.

SINUMERIK®-Documentation

Key to Editions

The Editions listed below have been published prior to the current Edition.

The column headed "Amendments" lists the amended sections, at all times with reference to the previous Edition.

Edition	Order No.	Amendments
08.88	6ZB5 410-1AA02-0BA0	New edition

Contents

1. Introduction

1.1 General

These Operating Instructions refer to SINUMERIK 3T and SINUMERIK 3M, in the basic versions 4 and SINUMERIK 3TT.
The individual controls are operated in the same manner.
The few situations where the operation of the controls differs, e.g. traversing with direction keys are described specifially for each control.

The operating concept of the System 3 controls is characterized by its systematically uniform operating logic.
The operation of the System 3 controls is facilitated and simplified by use of operator prompting (guidance).

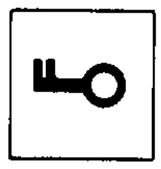
The operating steps are presented in order in which they occur, using, the symbols on the operating panel.
The Operating Instructions are arranged in the order the settings of the mode selector switch. Fold-out drawings of the operator panels of both types of control can be found in the Appendix.
The appropriate fold-out illustrations are useful aid to understanding the text.
The operator panel is divided into modules, which are also shown in the Appendix.

The individual operating elements are described as they appear in the text and the key number detailed on the fold-out drawing is indicated.

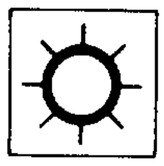
We reserve the right to change this description following any technical changes.

1.2 Symbol Description

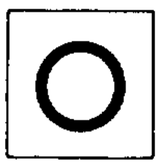
The following symbols are used in the Operating Instructions:



Key Switch Enabled



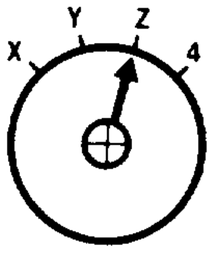
Lamp ON



Lamp OFF

Toggle Switch OFF

Toggle Switch ON



Rotary Switch, e.g. axis selector set to Z-axis
(for SINUMERIK 3M only)

Hand
Operation to be carried out only according to the condition indicated

X12345.678 Address with specified number of decades
(shown here: 8 decades, 3 decimal points)

4 4th. axis;
The name of the axis is defined by a machine datum
(SINUMERIK 3M only)

1.3 Terminology

Program File Indicator

The controller distinguishes between 10000 part programs.
A maximum of 200 programs can be stored at the same time.
The program pointer is used to call up and display the part program selected at the moment.

Example: program pointer % 1234

Program Status Pointer

The program status pointer displays the current operating status, and consists of:

- Block number in the part program	N 1234
- 1st. subroutine number with number of passes	L 123-1
- Subroutine block number	N 1234
- 2nd. subroutine number with number of passes	L13401
- Subroutine block number	N1234
- 3rd. subroutine number with number of passes	L14501
- Subroutine block number	N 1234

Select

Displays selection by key stroke (e.g. part program)

Cursor

Correction pointer

Page

CRT display page.

1.4 Operating Modes

The NC controls the movement of tools and workpieces on a machine tool, in accordance with a previously entered part program. Before the actual machining process can begin, a numerically controlled machine tool must be loaded with tools and workpiece, and the following preparations must first be completed.

These preparations are:

- moving the tool or the workpiece to the correct starting position as shown in the set-up drawings,
- loading the part program into the control memory,
- entering or checking the zero offsets,
- entering of checking the tool offsets.

During these preparations, the control must be put in the specific modes, so that the control can establish these preparatory operations. The operating modes can be selected with the mode selector switch on the machine control panel.

The different operating modes are:

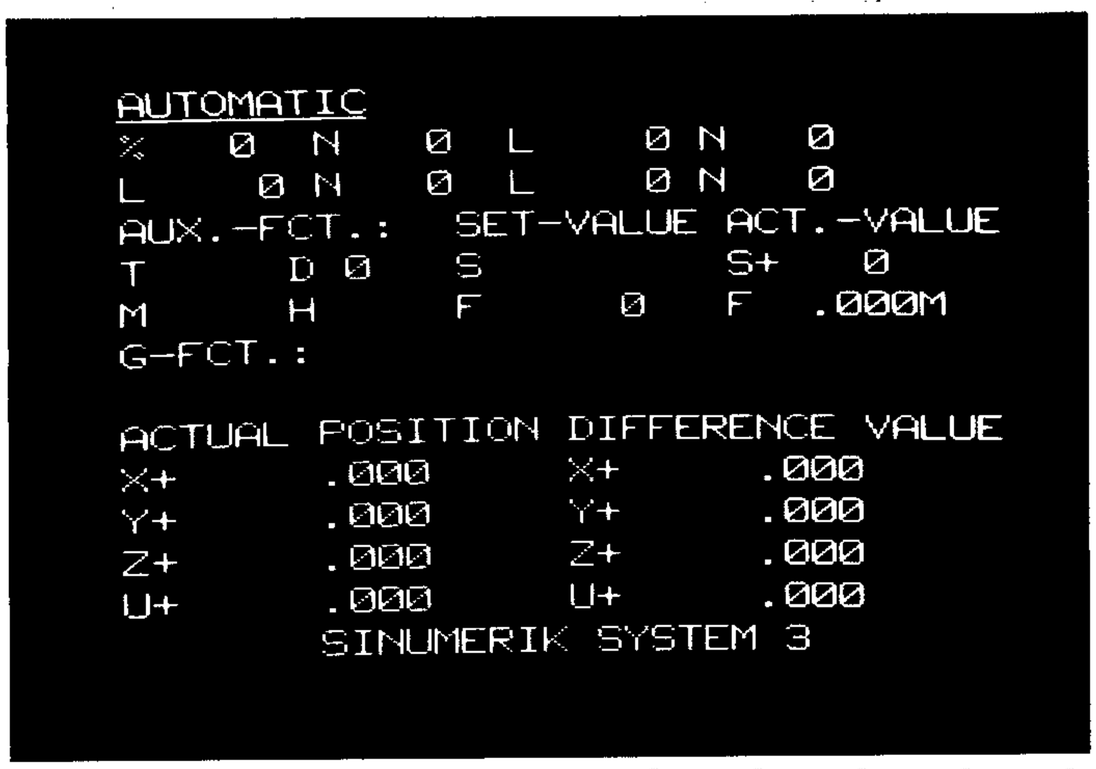
Symbol	Description	Display Text	Abbreviation
	Data output (interface)	DATA OUTPUT	DO
	Data input (interface)	DATA INPUT	DI
	Manual data input/ Automatic	MDA	MDA
	Jog, handwheel	JOG	JOG
	Incremental feed	INC FEED	INC
	Manual data input, part program	MDI PARTPROGRAM	MDI-PP
	Manual data input tool offsets zero offsets machine offsets Setting Data Testing Data	MDI SE-TE	MDI SE-TE
	Automatic	AUTOMATIC	AUT
	Reference Point Approach	REFERENCE POINT	REF

1.5 Input and Display List

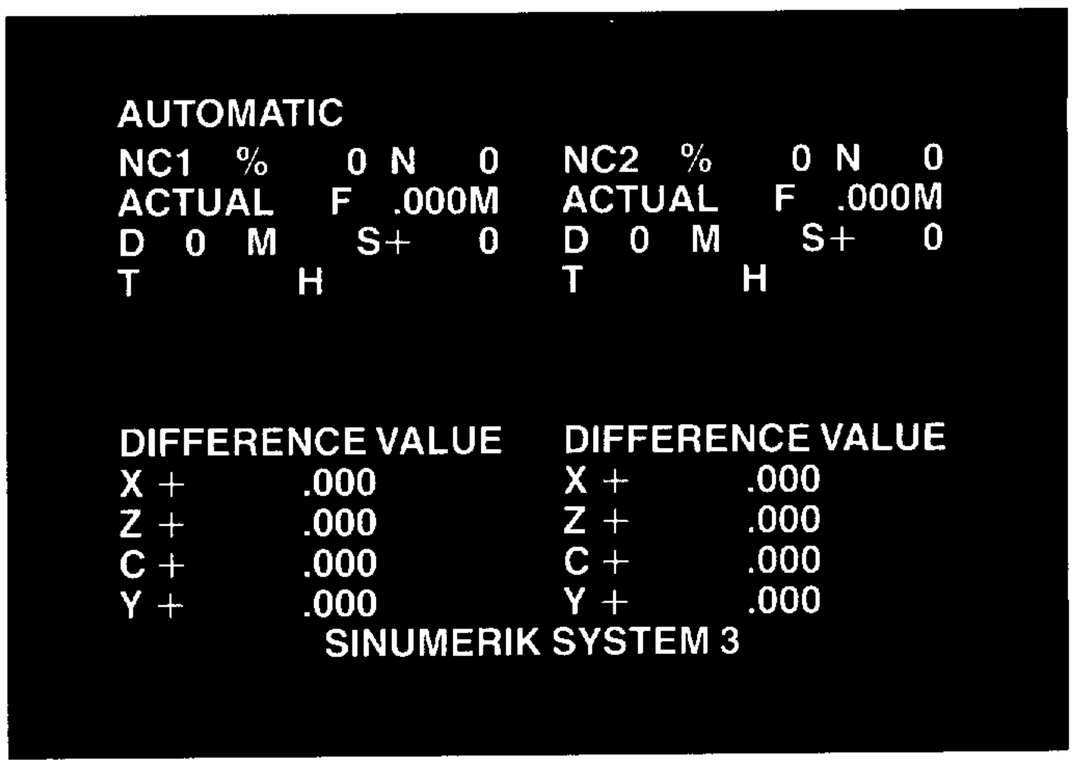
	Data output via data interface of part program %, subroutine L, tool offset data TO, machine data TE, R, H parameters			* Alarms 8.6.1 * Interface page 8.6.2 * Machine data 8.6.3	Data output via data interface 2. Data input via	* Position indicator 1.8
	Data input via data interface of part programm %, subroutine L, tool offset data TO, machine data TE, R, H parameters					
	*Basic display Input manual input block 4.1 Process manual input block 4.1 Store manual input block 4.2	* Tool offset data 8.1	* Zero offset 8.2 * User data 8.3/ 8.4 * R parameter 8.5 * H parameter	* Alarms 8.6.1 * Interface page 8.6.2 * Machine data 8.6.3	* Offset block 9.4 * Program indicator 9.4	* Position indicator 1.8
	Axis Traverse using direction buttons or handwheel 5.					
	Basic displays Overstoring of M, S, T, H functions 5.0	Automatic determination of tool offsets 8.1.7	Automatic determination of zero offsets 8.2.7	* Alarms 8.6.1 * Interface page 8.6.2 * Machine data 8.6.3	Playback 4.3	* Position indicator 1.8
	Incremental Jog using direction buttons 6.					
	Basic displays Overstoring of M, S, T, H functions 6.0	Automatic determination of tool offsets 8.1.7	Automatic determination of zero offsets 8.2.7	* Alarms 8.6.1 * Interface page 8.6.2 * Machine data 8.6.3	Playback 4.3	* Position Indicator 1.8
	Cancel part program % subroutine L 7.1	* Tool offset data 8.1	* Zero offset 8.2 * User data 8.3/ 8.4 * R parameter 8.5 * H parameter	* Alarms 8.6.1 * Interface page 8.6.2 * Machine data 8.6.3	Input,edit part programs % 7.3 subroutine L 7.4	* Position indicator 1.8
	*Basic displays	Input tool offset data 8.1	Input zero offsets 8.2 user data 8.3/ 8.4 R parameter 8.5 H parameter	* Alarms 8.6.1 * Interface page 8.6.2 * Machine data 8.6.3 * Drift offset 8.6.5	* Basic display	* Position indicator 1.8
	Basic displays Overstoring of M, S, T, H functions 9.0 Block search 9.5 * Program pointer 9.4 * Correction block 9.4	Processing a Part Program 9. Input tool wear and tool offsets 9.3	* Zero offset 8.2 * User data 8.3/ 8.4 * R parameter 8.5 * H parameter	* Alarms 8.6.1 * Interface page 8.6.2 * Machine data 8.6.3	Input, Editing 9.10 Part programs % 7.3 Subroutines L 7.4	* Position indicator 1.8
	Reference Point Approach using direction buttons 10.					
	* Position indicator 1.8			* Alarms 8.6.1 * Interface page 8.6.2 * Machine data 8.6.3		* Position indicator 1.8

Note : * indicates only displays,data input is not possible.
The numbers indicate the sections in the manual which deal with these functions.

1.6 CRT



"Automatic" basic display 3M



"Automatic" basic display 3TT
(double-height display option)

The display structure is largely identical in all operating modes. The monitor has 16 lines of 32 characters each.

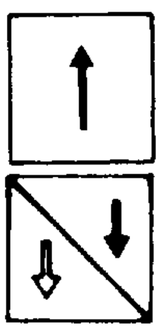
The arrangement of the displays (shown here - Automatic display 3M) is generally indentical for all the operating modes. The CRT has 16 lines of 32 characters each.

The 1st. line displays the operating mode selected.

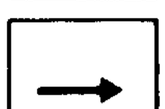
The 14th. line displays the first NC alarm number with its alarm text.

The last 15 characters of the last (16th.) line are reserved as input line.

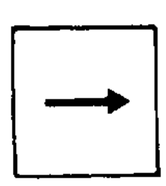
Since the CRT displays a large amount of data, the following keys are used to search for specific information:



PAGE keys
Scroll Down, key No. 22, and Scroll Up, key No. 23)
(when displaying part programs incrementing block-by-block).



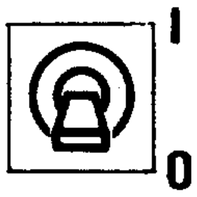
and the



Cursor keys
(cursor left, key No. 24, and cursor right, key No. 25)

1.7 Control Power-on

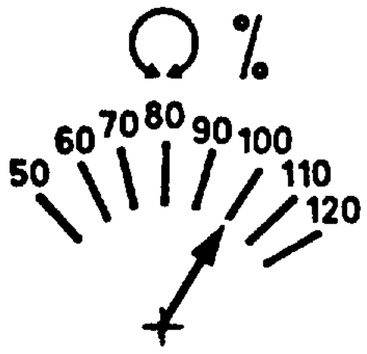
Before the control is switched on for the first time, it is important to ensure that the control was started up by qualified personnel.
Before starting, set the switches to the following positions:



Single block (switch No. 40)

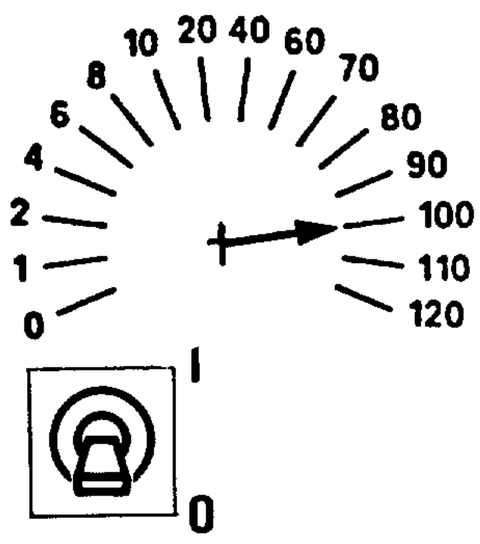
Single delete (switch No. 41)

Dry run (switch No. 42



Spindle speed override switch (No. 32) 100%

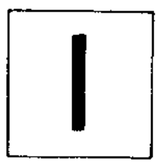
Feedrate override switch (No. 33) 100%



Rapid traverse override active (No.43)

In the following it is assumed that input and programming were done in the metric system.
The feedrate displayed is in mm/min.

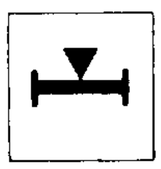
Power ON



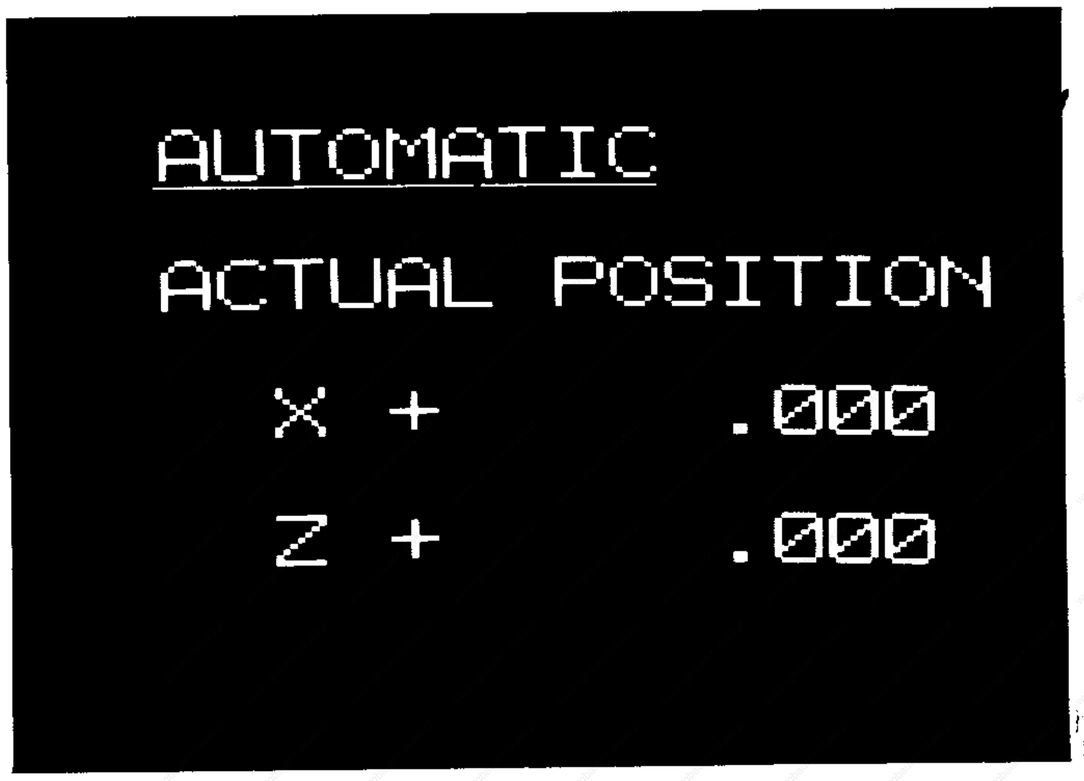
After the control has been switched on (key No. 37), the basic display of the current operating modes appears (see the Input and Display List, Section 1.5)

1.8 Actual Value Display

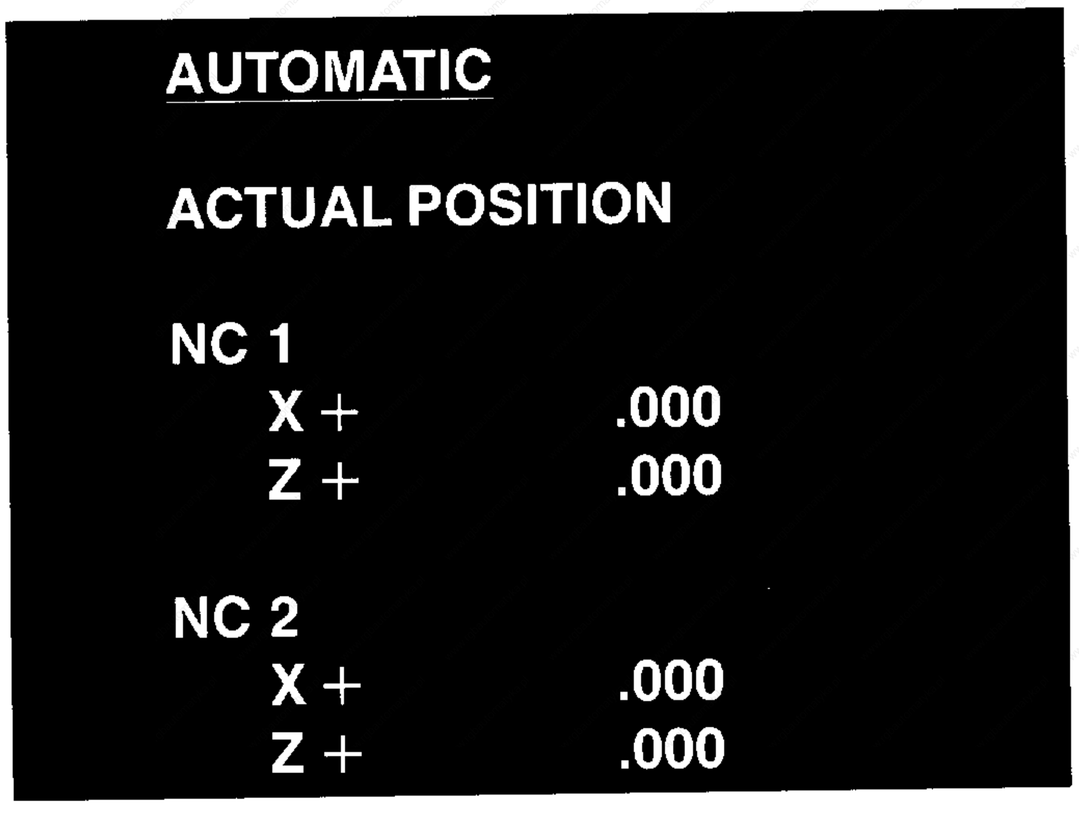
The position display in double-height characters can be selected in any operating mode.



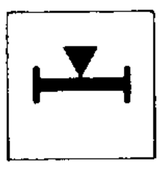
Select position display (key No. 18)



SINUMERIK 3T



SINUMERIK 3TT/dual slide display

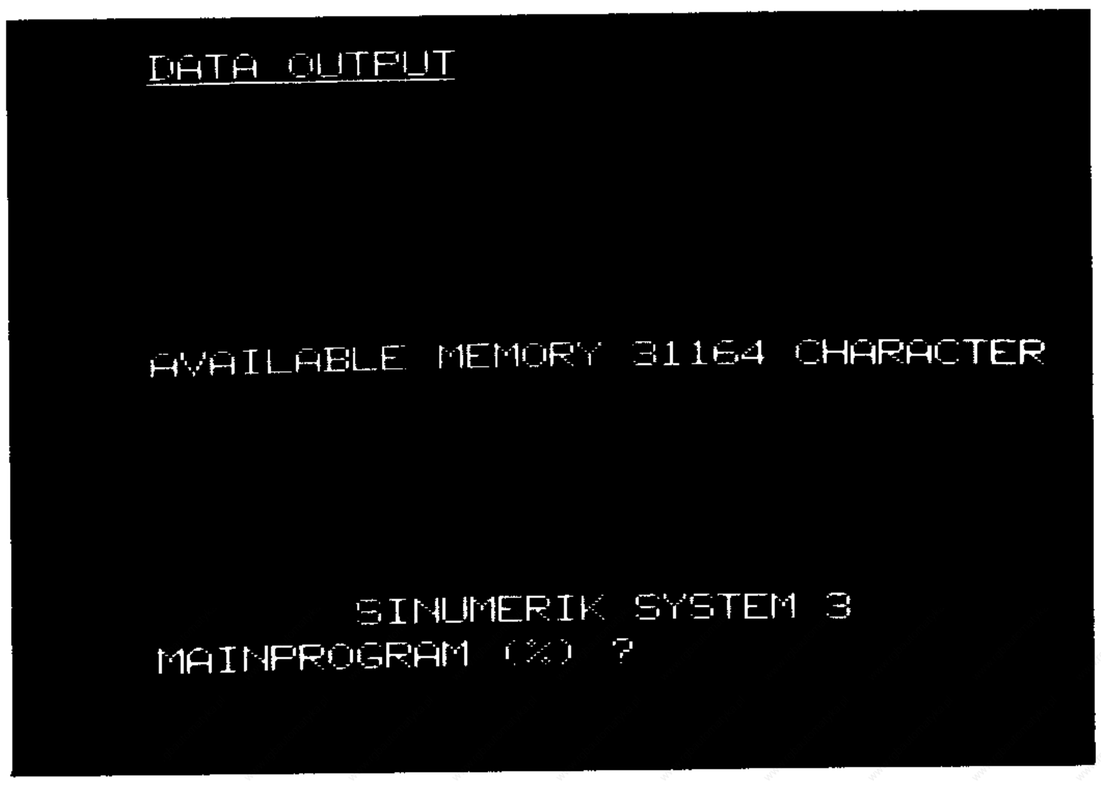


Deactivate (key No. 18)

After the position display in double-height characters has been deactivated, the basic display of the current operating mode reappears.

2. Data Output (DO)

After the operating mode has been selected, the following display appears:



Line 6: Available part program memory
(Tape characters = ASCII characters)

Line 15: Operator prompting for selection of output data

The following data can be output:

Main programs	%
Subroutines	L
Tool offset	TO
R parameter	R
Background memory	H
Machine data	TE

Operation Sequence

Operating mode data output (mode selector switch No. 31)

NO

Selection of the data type by paging with the key (No. 27). These are displayed in the sequence %, L, TO, R. H and TE in the 15th line of the display.

YES

The data in line 15 is selected for output by pressing this key (No. 26)

Example: Output of a single part program

% 1234

only with %, L, R and H

Input of the part program number and termination with the input key (No. 16)

```
END ?
```

NO

By activating the NO-key (No. 27), the part program number is automatically completed for the end of the output.

```
DATA OUTPUT

  % DATA ARE BEING GIVEN OUT !

    BEGIN :  %1234
    END   :  %1234

AVAILABLE MEMORY 31164 CHARACTER



          SINUMERIK SYSTEM 3
  START !
```

Start data output (key No. 28)

The text CONTROL BUSY is displayed in line 15 during data output.

Example: Output of several part programs

% 1234 only with %, L, R and H

Input of the first part program number and termination with the input key (No. 16)

```
% 1500
END ?
```

Input of the last part program number and termination with the input key (No.16)

```
DATA OUTPUT

 % DATA ARE BEING GIVEN OUT !

   BEGIN : %1234
   END   : %1500

AVAILABLE MEMORY 31164 CHARACTER


        SINUMERIK SYSTEM 3
START !
```

Start data output (key No. 28)

CONTROL BUSY is displayed during data output in line 15.

Special case

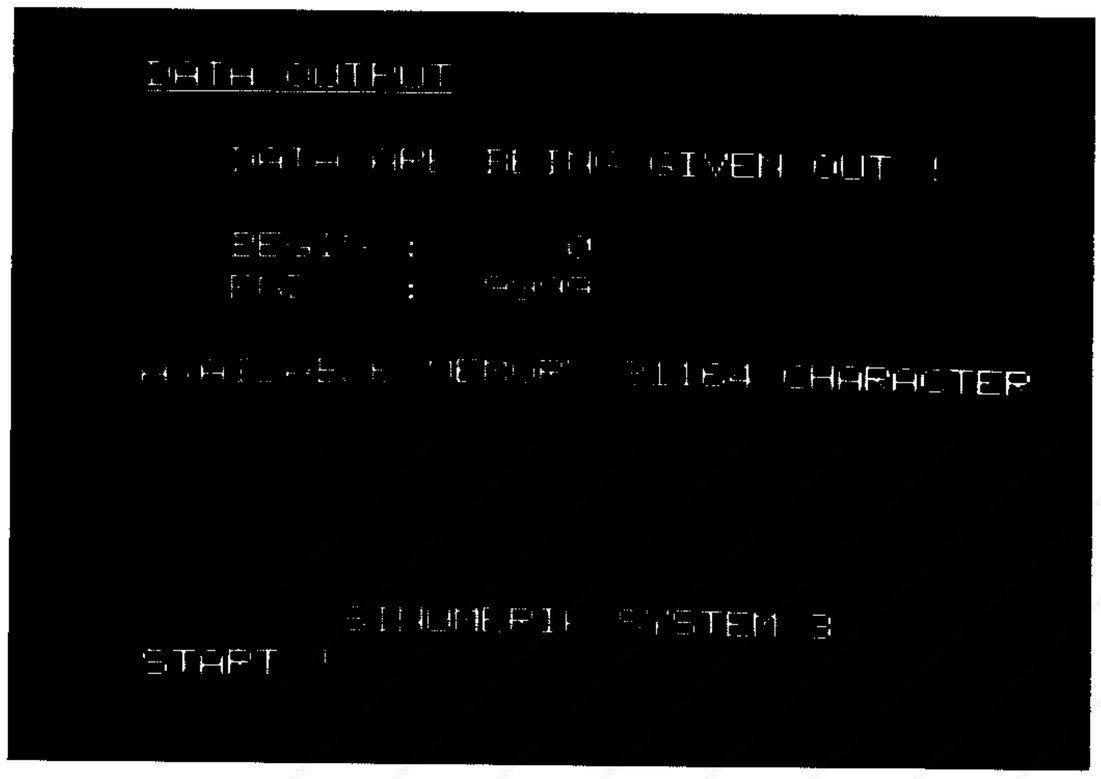
Output of measuring data protocol stored in the control as a subroutine. This output must be triggered from the PLC with "data start".

Example: Output of all part programs

NO

only with %, L, R and H

The part program %0 through %9999 are selected automatically by activating the NO key (No. 27)



Start data output (key No.28)

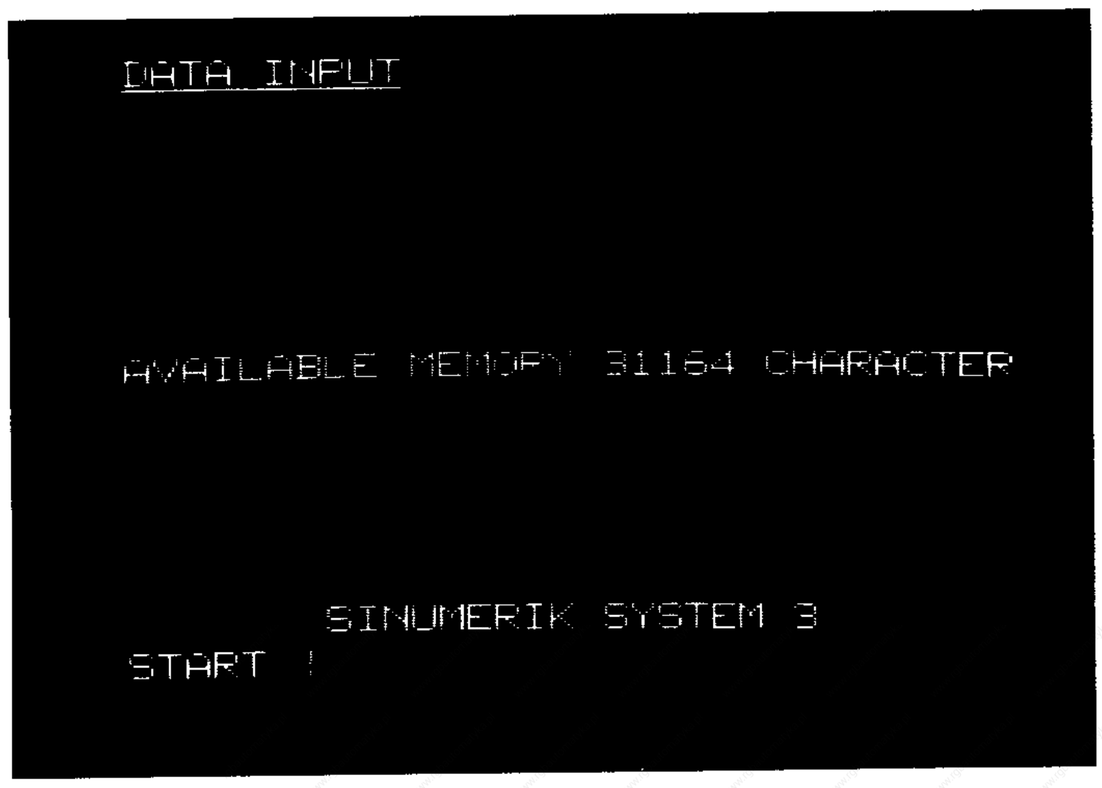
CONTROL BUSY is displayed in line 15 during data output.

One data type is output per operator input. In part programs it is also possible to output individual part programs. Individual R parameter values and background memory values (H parameter) can also be output. Output code (ISO or EIA) is defined in the user data.

At the beginning and end of the program a leader tape of approx. 0.3 m with transport holes is output. Between individual main programs and subroutines approx. 5 cm of transport holes are also output (option). If a longer leader tape is required, this must be punched specially.

3. Data Input (DI)

The following display appears:



Line 6: Still available part program memory
(Tape characters = ASCII characters)

Line 15: Operator prompting for initiating data input

The following types of data can be input:

Data type	Screen display	Punched tape identifier
Main program	%	% ... LF
Subroutine	L	% SP LF
Tool offset	TO	% TO LF
R parameter *)	R	R% R LF
H parameter *)	H	% H LF
Machine data *)	TE	% TE LF

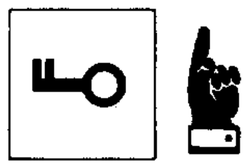
*) Can be interlocked with a switch

The data type is recognized automatically during reading.

Operation Sequence

Operating mode input (mode selector switch No. 31)

Key switch (No. 38) enabled

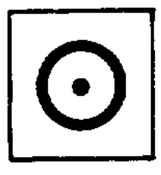


(dependent on machine datum)

```
DATA INPUT

DATA ARE BEING READ IN

AVAILABLE MEMORY 31164 CHARACTERS

    SINUMERIK SYSTEM 3
CONTROL BUSY
```

Start data input (key No. 28)

During data input, line 15 displays CONTROL BUSY, which disappears upon completion of data transfer.



Alarm 238 (in uncontrolled data transmission) appears if data are not transmitted by the reader within 20 s. This time monitoring can be switched off via user data.

During data input, reading is continuous up to the end criteria M02 or M30.

3.1 Checking the Input Data

The input data are checked for simple errors.

- Character parity
  In the ISO code every character must have an even number of bits (logic "1"), and in the EIA code every character must have an odd number of bits. (This corresponds to the number of holes per character on the tape)

- Block parity
  The number of characters in a block (including LF) must be even.
  Block parity checks can be activated or deactivated via the user data.

- Double read in
  If the same program (identical program number) is read in twice, the newly read program is compared with the stored program.

If any error is recognized, the reading is stopped and alarm 277 is displayed.

3.2 Interface and Interface Devices

An interface with the following specification is available for data transfer:

See also the "SINUMERIK System 3 and 8 Universal Interface for Data Input and Output" Description.

Full duplex * { 20 mA current loop
RS 232C (V.24); in accordance with DIN 66020

The interface data for input and output can be set separately using machine data.

Inputs and outputs are only possible to and from an internal memory.

The following devices can be connected:

reader, punch, combined reader-punch device,
e.g. Siemens Printer PT 80,
as well as other devices that comply with the interface specifications.

* See also "SINUMERIK System 3 and 8 Universal Interface for Data Input and Output" Description.

4. Automatic Mode with Manual Input (MDA)

This operating mode allows three operating methods:

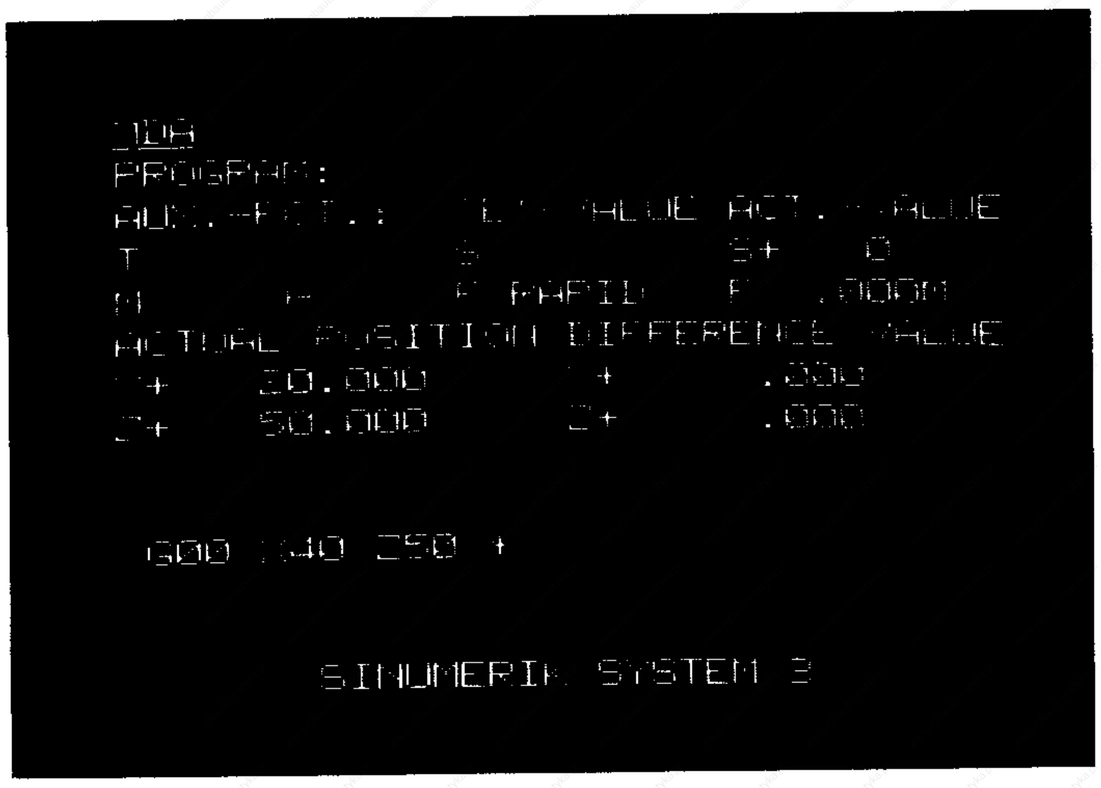
- Manual data input
- Teach-in
- Playback

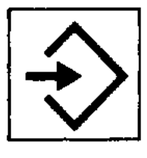
I Single block (switch no.40)
0 must be set to OFF (0).

4.1 Manual Data Input

Direct manual input and execution are possible.

Operating mode MDA (mode selector switch no. 31)





Block input word by word including LF, according to the normal data input rules.
The block entered is shown in the display.

The block is executed by pressing the Cycle Start key (No. 44).
It remains displayed.

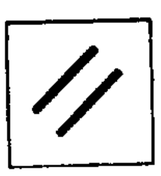
The executed block can be cancelled with the "Cancel" key
(No. 14)

The next block can now be entered.
The following blocks are not permitted:
Cutter radius compensation (CRC), or tool nose radius compensation (TRC),
and contour blocks.
These blocks can be written in a subroutine and called in MDA mode.

If the mode selector switch (No. 31) is switched from MDA to AUT or
vice-versa, a reset occurs automatically.

4.1.1 MDI Termination

If during the processing of a block an error is found, that block can be terminated.



Reset (No. 12)

All the remaining programmed paths are cancelled.

4.1.2 MDI Editing

The input line can also be deleted character-by-character if this function is selected via machine data.

The block entered (but not yet executed) can be completely cleared with "Cancel" (No. 14).

New data input through the input line.

4.1.3 MDI Interruption

MDA can be interrupted by using the operating modes, JOG, INC, or MDI-PP. This corresponds to "Automatic interrupted".

The bottom line displays:

When returning to MDI, paths traversed with JOG or INC are taken into account.

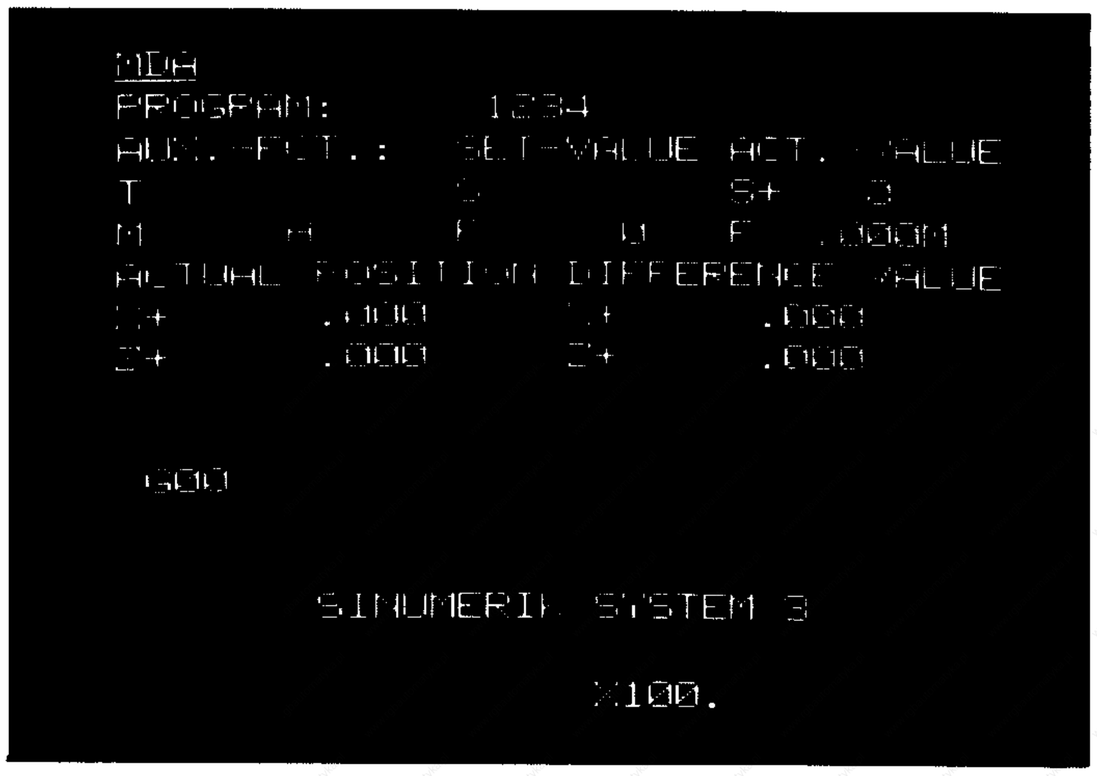
If an MDI block is interrupted, the paths traversed with JOG or INC are calculated and the programmed end point is approached.

Upon continuation of MDI, the next traversing block is referred to the last MDI block.

4.2 TEACH-IN

MDI blocks executed as described in 4.1 can be consecutively transferred into the program memory.

Operating mode MDA (selector switch No. 31)



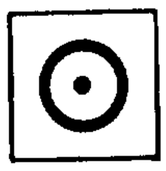
A program number is defined with the input key (No. 16) at the beginning of the program.

The "Cycle Start" key (No. 44) decodes the program number for the part program memory.

The program is started by pressing the "Start" key (No.28)

Word-by-word input of the block including LF, following the usual data input rules. The block entered is displayed.

The entered block is executed by pressing the "Cycle Start" key (No. 44). It remains displayed.
If "Playback" (see Section 4.3) has been previously selected, the "Cycle Start" key (No. 44) must be pressed twice.



The executed program can be transferred into the part program memory by activating the "Start" key (Nr. 28). After the transfer, the block disappears from the display. The next block can now be entered.

If the program has been programmed only in absolute dimensions (G90), an incorrectly processed block can be omitted (no transfer with the "Start" key) and only the next block with the correct final position is transferred.

Stored blocks can be edited in the MDI-part program operating mode (see Section 7). In each case only the last program entered is associated with the MDI automatic mode.
After an interruption caused by switching modes, the program can be continued in MDA part program. However, if reset was in effect (e.g. data input), all modal functions must be re-entered in the first block of the program to be continued.

4.2.1 TEACH-IN Interruption

(see also 4.1.3)

4.3 PLAYBACK

In the setting-up mode, the positions approached are stored as command values in the program memory.
Missing feedrates, switching and auxiliary functions can be added by using the program edit function.

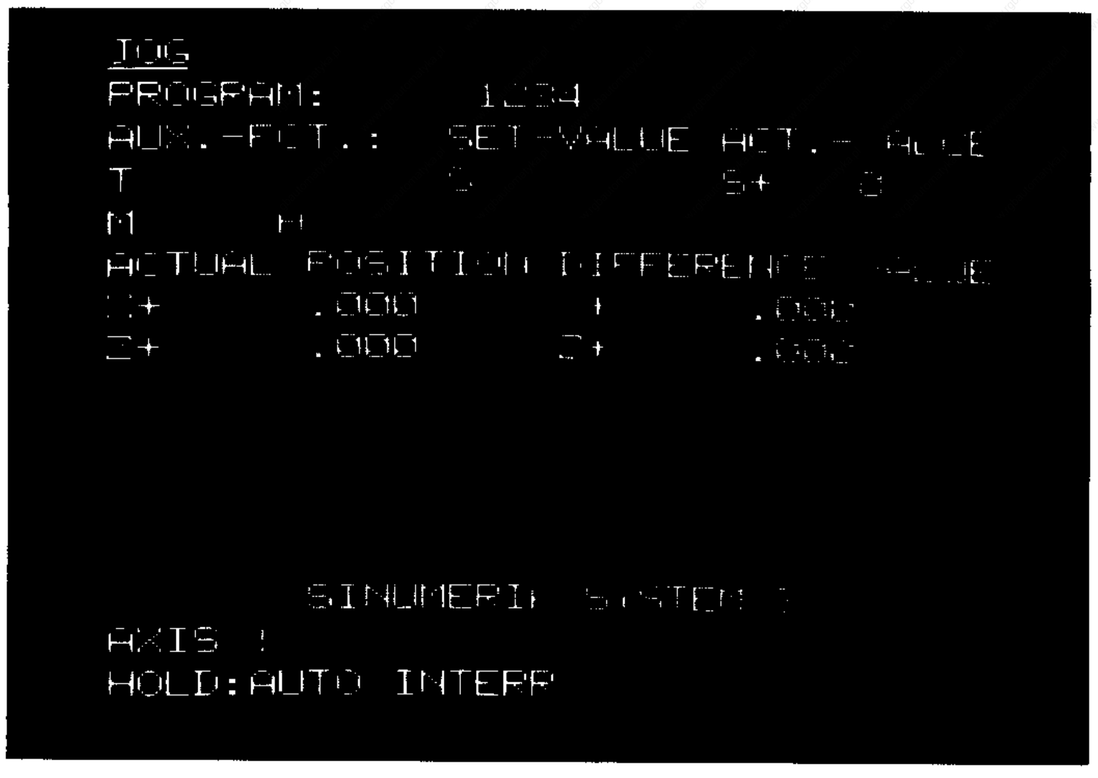
Start program in teach-in
Mode selector switch to MDA (mode selector switch No. 31)

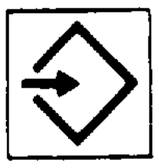
e.g. % 1234 LF input key (No. 16)

Press the "Cycle Start" (No. 44) and the "Start" (No. 28) keys to start a program.

Select set-up modes JOG or INC by using the selector switch (No. 31).

The following page is displayed:





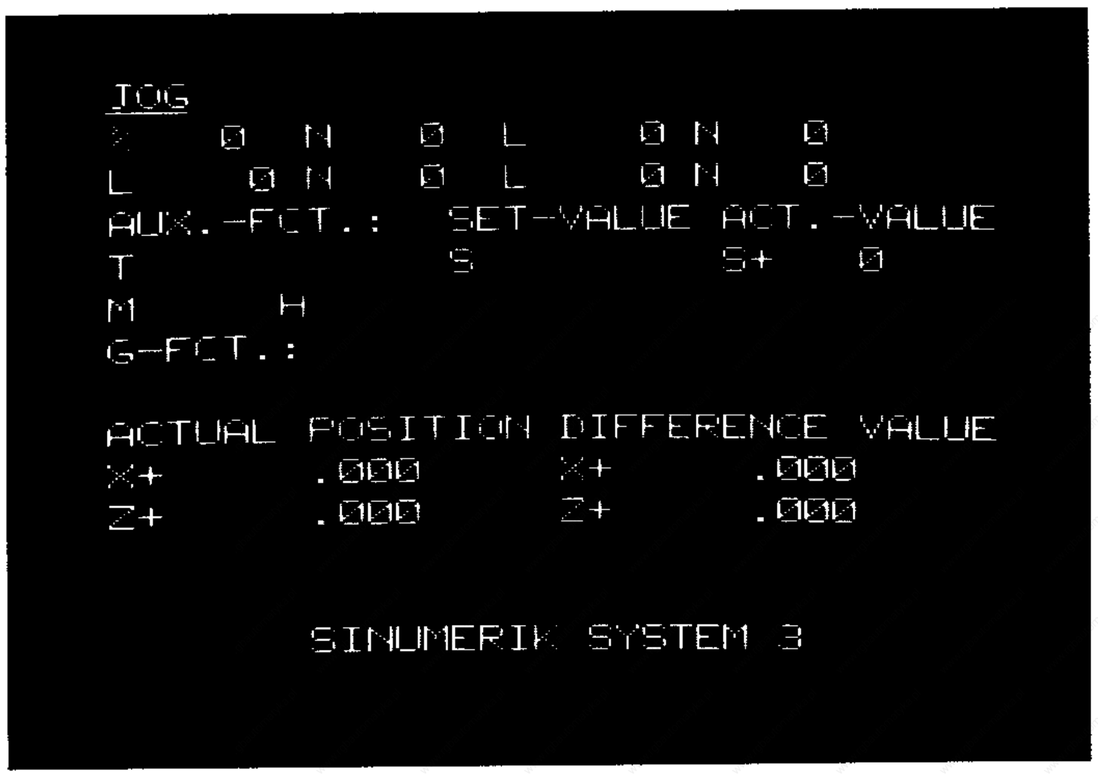
Coordinates reached in the JOG mode are stored as program blocks (linear interpolation), by reading the appropriate axis addresses, e.g. X, and by pressing the Input key (No. 16) and the Start key (No. 28).

G01 must be active from a program block already entered in MDA-TEACH-IN.

5. JOG (Handwheel)

Operating mode JOG (selector switch No. 31)

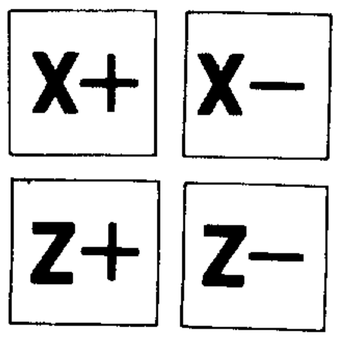
The following display appears:



In this operating mode the S, M. H, and T functions can be transferred (overstore) (see Section 9).

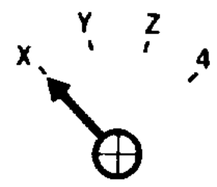
5.1 Feed and Manual Rapid Traverse

For the 3T

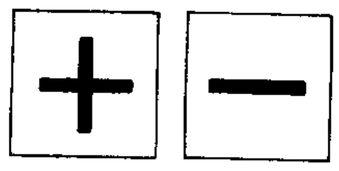


Direction keys (No. 34)

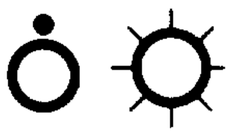
For the 3M



Axis selection switch (e.g. X) (No. 47)

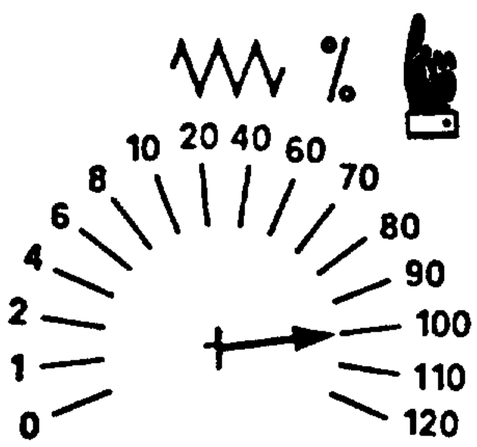


Direction keys (No. 34)

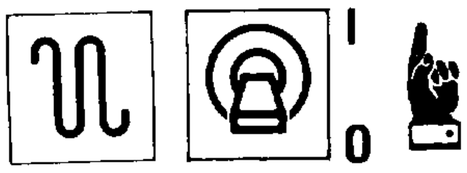


Feed Stop (No. 2) should not be active

The rapid traverse override key (No. 35) can be pressed for rapid traverse.



The feed speed can be modified with the feedrate override switch (No. 33).



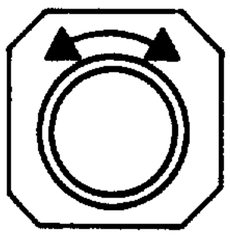
If traversing at rapid traverse with the toggle switch (No. 43) in the "rapid traverse override" position, the rapid traverse override (No. 33) is active within a range from 0 to 100%.

The position 0 % on the feedrate override switch leads to stopping during feed or rapid traverse, regardless of the position of this switch.

5.2 Handwheel Operation

The handwheel moves the axes in the same manner as JOG.

Handwheel movement is selected, just like "Feed and Rapid JOG", in the JOG operating mode (mode selector switch No.31)

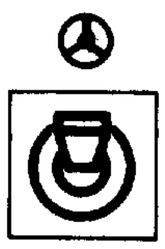


Only one handwheel can be connected.

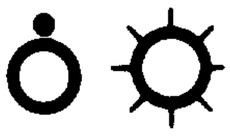
For 3 M

Handwheel is assigned to an axis by using the axis selector switch No. 47

For 3 T



Handwheel assigned to axis using toggle switch No. 46



Feed hold (No. 2) should not be active

When a handwheel has been installed, both it and the direction keys can be used for jog operation. If the handwheel and the direction keys are used simultaneously, the keys take precedence.

The handwheel can be used for $^{0}/_{um}$ (handwheel inactive), $^{1}/_{um}$. $^{10}/_{um}$ and $^{100}/_{um}$ per scale division.

The scale valuation is set through the "bit oriented user data" (see Appendix).

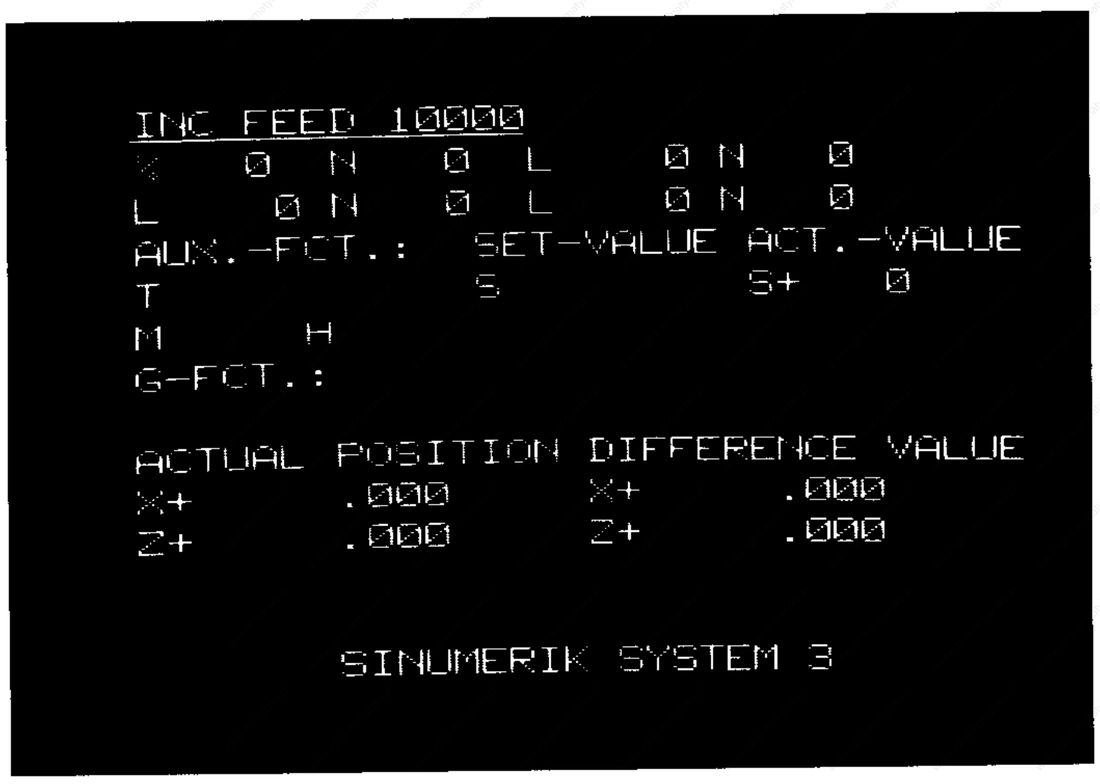
Mixed operations (1 axis assigned to handwheel, 1 axis assigned to direction key) are not possible.

6. Incremental Mode (INC)

In the absence of a handwheel, the incremental jog mode can be used to move away from a contour or for accurate setting-up.
Manually defined paraxial movements are possible.

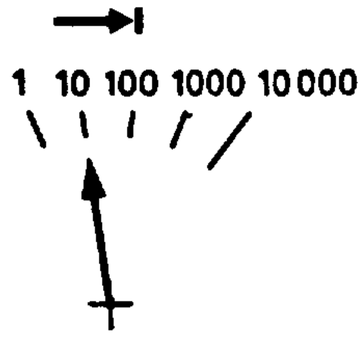
Feedrate F is defined via machine data and is dependent on the feedrate override switch (No. 33).

→| INC mode (mode selector switch No. 31)



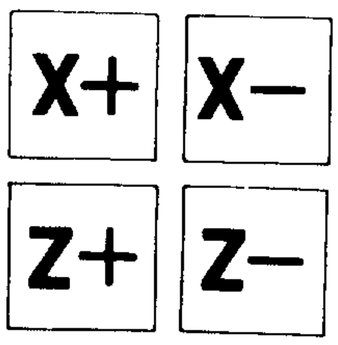
M, S, H, and T functions can be transferred (overstored) in this operating mode.

Operating Sequence



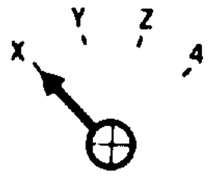
For the 3 T

Preselect the increment size using the mode selector switch (No. 31)

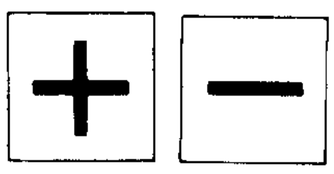


Press the desired direction key (No. 34)

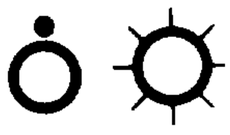
For the 3 M



Axis selector switch (e.g. X) (No. 47)



Direction key (No. 34)

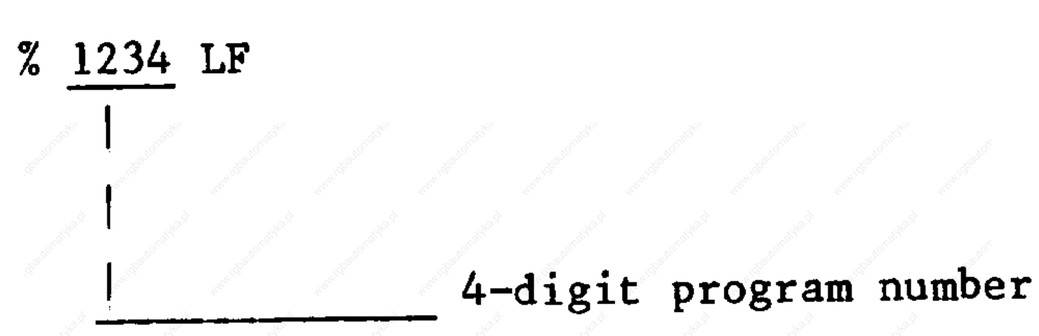


The feed hold (No. 2) should not be active.

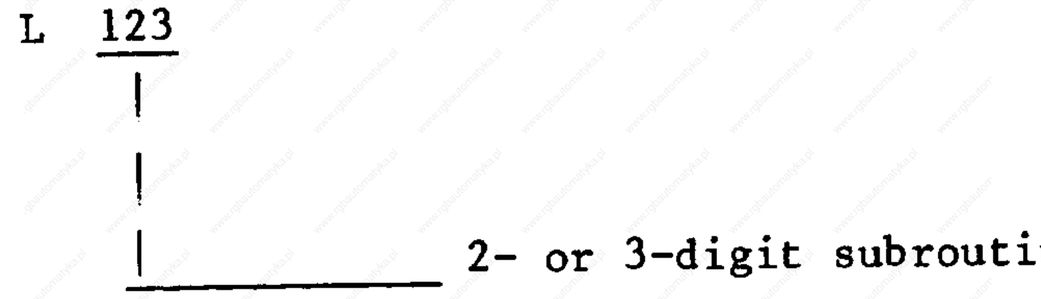
The incremental step movements are executed modally.

7. Manual Data Input, Part Programs

Part programs are always stored in the part program memory, and are processed from there. The NC assigns the memory areas for part programs and subroutines automatically (operator inputs are not required). Up to 200 part programs and subroutines can be simultaneously stored in the memory. Part programs can be numbered and have the following identification:



Subroutines are identified by a 2- or 3-digit number:



Any part program without a program number is assigned a "0" by the NC, and this is neither displayed nor punched out. Only one part program without program number can be stored in the program memory.

Operation Sequence

Mode selector switch (No. 31) on MDI-PARTPROGRAM

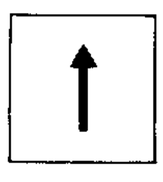
The following display appears:

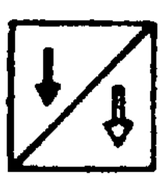
```
MDI PARTPROGRAM
PROGRAMM:     % 100
 %100 *
 N5 G00 G91 *
 N10 X100 Z0 *
 N15 X90 Z5 *
 N16 X20 Z10 *
 N20 R00 124.55 *
 N25 S1000 M03 *
 N30 M55 *
 N35 G01 F2000 X80 Z100 *
 N40 T1212 *
 N45 M30 *
        SINUMERIK SYSTEM 3
```

The CRT shows programs already stored (here % 100)
(EOB character LF = *)

If the mode is switched to the basic display (//) No. 5 using Display Changeover key (No. 17), a list of the stored programs appears.

```
MDI PARTPROGRAM
PROGRAM LISTING
%    1     88CH    %    2     82CH
%   96      4CH    %   94      4CH
%7777     25CH    %   10      4CH
%    5     88CH    %4222     15CH
L   94    935CH    L   96    451CH
 8888     10CH     4564     10CH
%9999     28CH    %  741      5CH
%8541      6CH    %  321      5CH
L  123     7CH    L  999      7CH
L  101     7CH    %  100    110CH
AVAILABLE MEMORY 31160 CHARACTER
        SINUMERIK SYSTEM 3
```





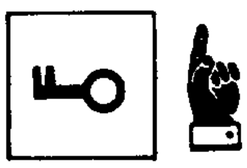
Using the keys 22, 23, ten displays containing the following information can be selected:

- unassigned memory space (here 31160)
- part program available (with associated program lengths)
- subroutine available (with associated program lengths)

Inputs are not possible into this display, only deleting and copying of programs.

7.1 Deleting and copying programs

- Deleting



Key switch enabled (No. 38)

dependent on machine datum

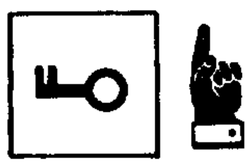
Switch mode to basic display

% 5

Input the program number and press the "Cancel" key (No.14). The program is deleted. Subroutines are deleted in the same way (e.g. L5 cancel).

By entering %- and pressing the "Cancel" key (No. 14), all the main programs can be deleted.
By entering L- and pressing the "Cancel" key (Nr. 14), all subroutines except L80 through L99 and L900 through L999 are deleted. These remaining subroutines are deleted only if the "Cycle lock" interface signal is not present.

- Copying



Unlock key operated switch (No. 38)

Depending on the machine data, switch mode to basic display

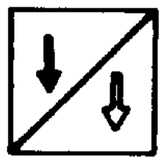
%12.15

Copying is started by entering the number of the program to be copied (source) and the program number of the new program (target) and pressing the "Edit" key (No. 15). The following can be copied.:
Main program to main program %12.15
Subroutine to subroutine L12.340

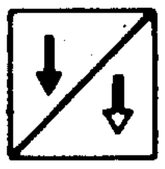
7.2 Program Select

Switch mode to part program input (No. 31)

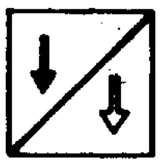
% 50 Select program, e.g. program % 50

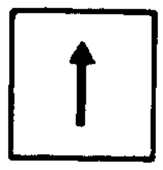


N15 Select block, e.g. block N15

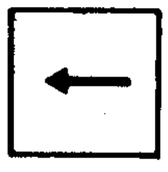


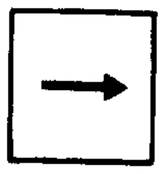
Before the desired block number is selected, the appropriate program number must have been selected.





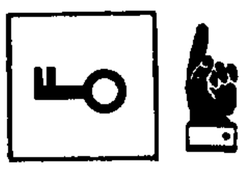
The display will change one block at a time when the "page" keys (page forward No. 22, page back No. 23) are depressed.





The "Cursor" keys (No. 24, 25) shift the cursor by one program word.

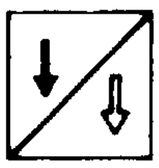
7.3 Keyboard Data Input to Memory



Keyswitch enabled (No. 38)

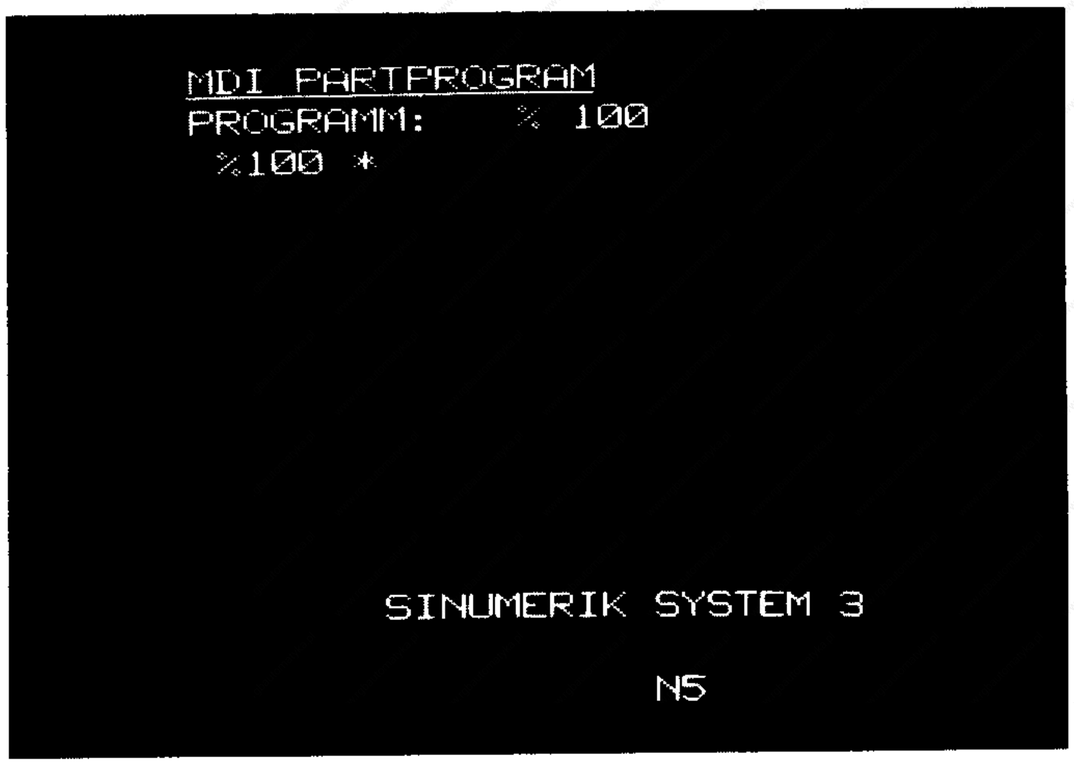
dependent on machine datum.

% 100



To input a new program, a program number, which is currently not in the memory (in this case % 100) is selected. If the % 100 program has already been stored, it is displayed. The cursor is positioned after the first program number.

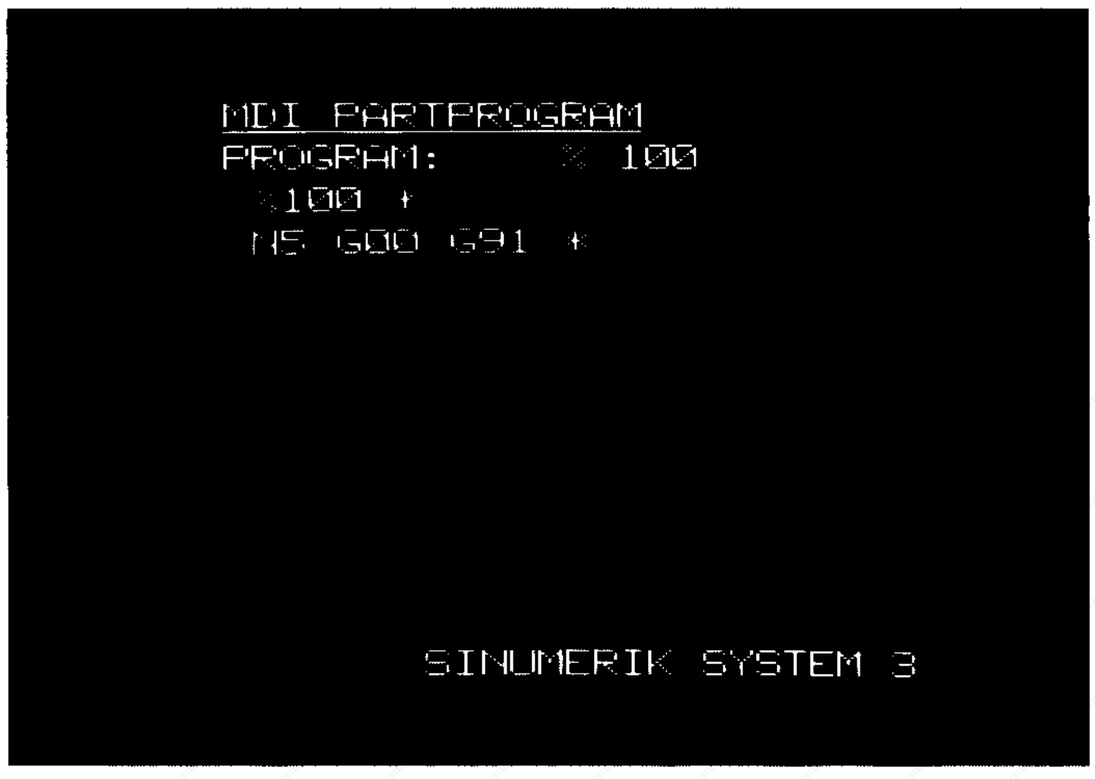
If the % 100 program is not stored in the memory, the first block "% 100 LF" is stored automatically and displayed on the CRT. The cursor is positioned behind the LF. The program blocks can now be entered via the operator panel.



Example:

N5 G00 G91 LF Enter the information, one word at a time

and press the "Input" key (No. 16) after each word.



If automatic block number default is selected (via machine data), a new block number is produced in increments of 5 in the input line after each block has been completed.

When entering or correcting programs, the block length (maximum 120 characters) and the amount of free program memory is checked. Incorrect conditions will trigger an alarm.

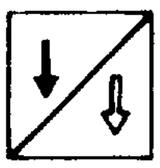
In the event of an error, the input line is rejected, the transfer of data will be prevented.

7.3 Keyboard Data Input to Memory

Keyswitch enabled (No. 38)

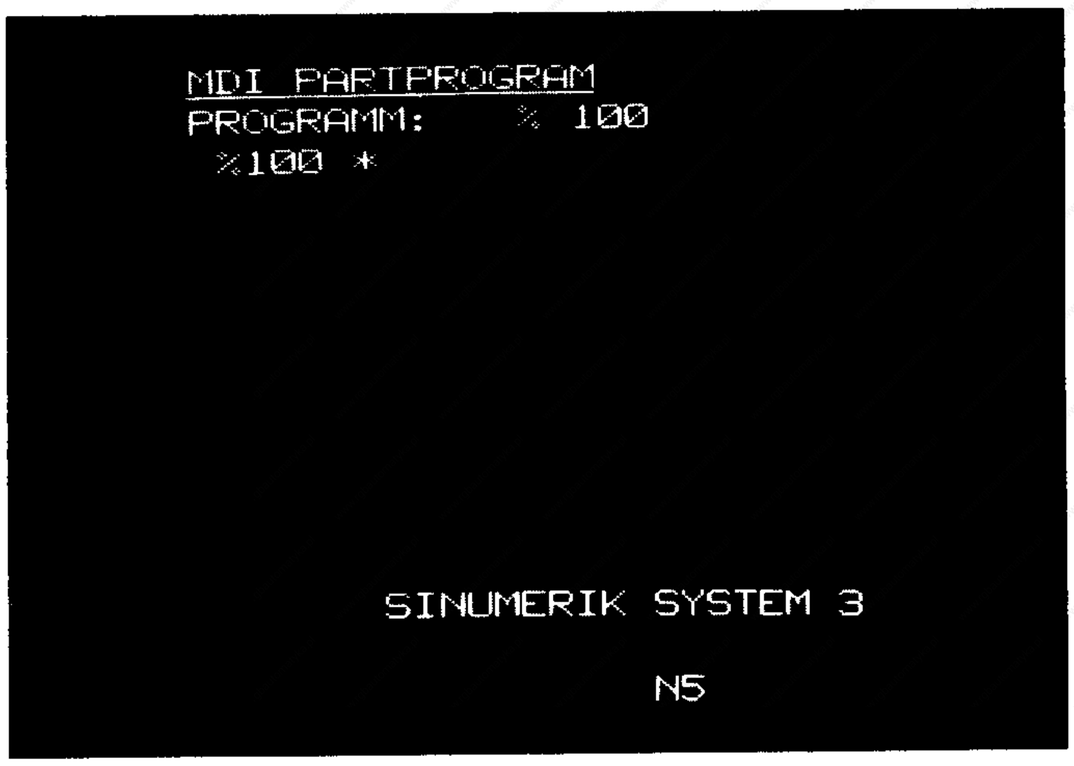
dependent on machine datum.

% 100



To input a new program, a program number, which is currently not in the memory (in this case % 100) is selected. If the % 100 program has already been stored, it is displayed. The cursor is positioned after the first program number.

If the % 100 program is not stored in the memory, the first block "% 100 LF" is stored automatically and displayed on the CRT. The cursor is positioned behind the LF. The program blocks can now be entered via the operator panel.



Example:

N5 G00 G91 LF Enter the information, one word at a time

and press the "Input" key (No. 16) after each word.

If automatic block number default is selected (via machine data), a new block number is produced in increments of 5 in the input line after each block has been completed.

When entering or correcting programs, the block length (maximum 120 characters) and the amount of free program memory is checked. Incorrect conditions will trigger an alarm.

In the event of an error, the input line is rejected, the transfer of data will be prevented.

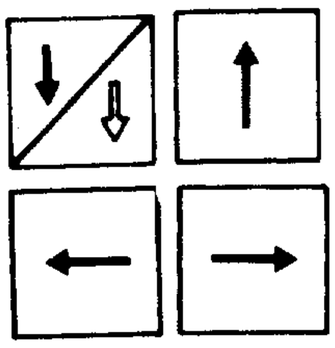
7.4 Program Editing

Keyswitch enabled (No. 38)

dependent on machine datum.

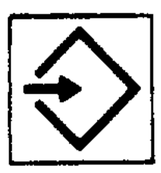
7.4.1 Inserting data

In the program % 100 entered previously, we want to insert the block N16 X20 Z10 LF.
Select the program as described in Section 7.2.

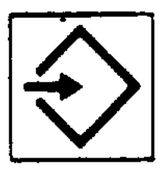


The "Page" and "Cursor" keys are used to bring the cursor (correction pointer) in front of the N20 block.

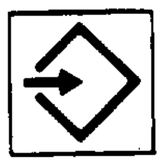
X20



Z10



LF



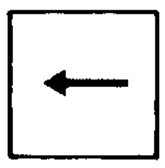
Block N16 is inserted (see Section 7.3)

```
MDI PARTPROGRAM
PROGRAMM:    % 100
 %100 *
 N5 G00 G91 *
 N10 X100 Z0 *
 N15 X90 Z5 *
 N16 X20 Z10 *
 N20 R00 124.55 *
 N25 S1000 M03 *
 N30 M55 *
 N35 G01 F2000 X80 Z100 *
 N40 T1212 *
 N45 M30 *
        SINUMERIK SYSTEM 3
```

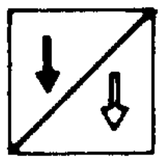
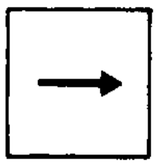
7.4.2 Modifying Single Words

In block N16, the value Z1O should be changed to Z20.

Select the program as described in Section 7.2.

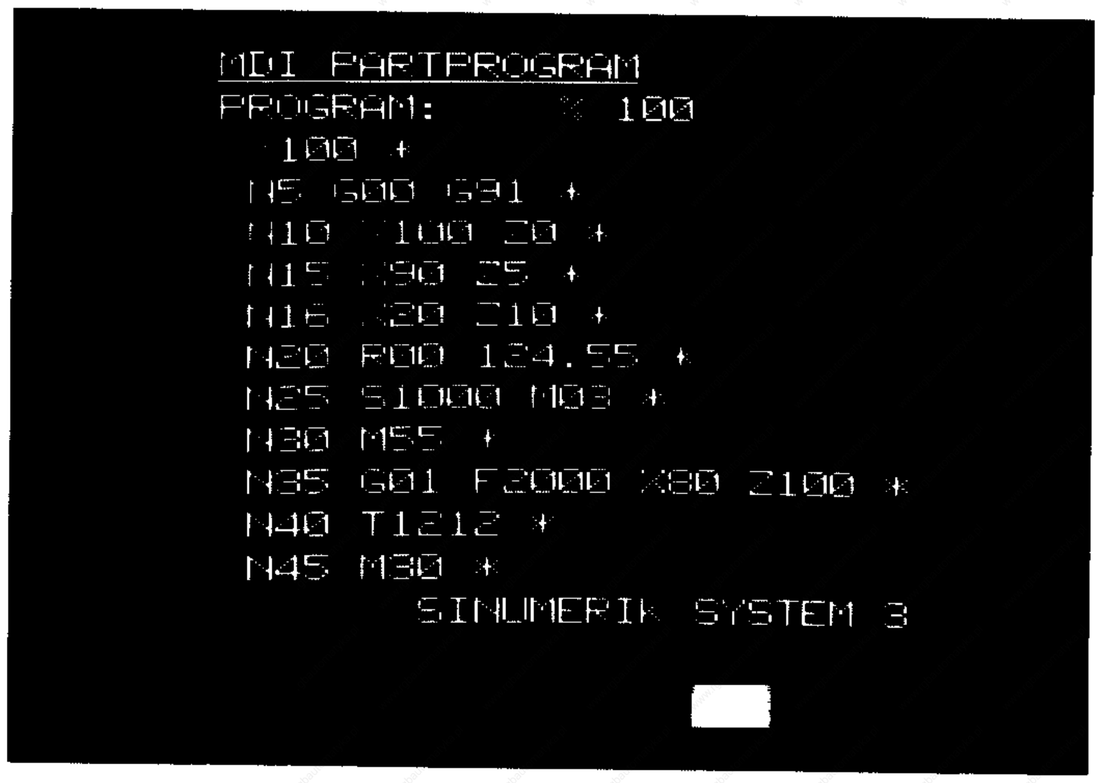



The Page and Cursor keys (No. 22, 23, 24, 25) are manipulated to bring the cursor in front of the word to be changed; in this example, Z1O.

X2O. Example of an incorrect entry: X2O entered instead of Z2O,

and completing the input with the "Edit" key (No. 15)



The faulty entry (when the address entered does not coincide with the address to be altered) is signaled by flashing in inverse video.
This information is transferred to memory.

The "Clear" key (No. 29) clears the faulty entry.

Z20. Input of correct value Z20

```
MDI PARTPROGRAM
PROGRAMM:    % 100
 %100 *
 N5 G00 G91 *
 N10 X100 Z0 *
 N15 X90 Z5 *
 N16 X20 Z10 *
 N20 R00 124.55 *
 N25 S1000 M03 *
 N30 M55 *
 N35 G01 F2000 X80 Z100 *
 N40 T1212 *
 N45 M30 *
        SINUMERIK SYSTEM 3

                        Z20
```

Input completed with the "Edit" key (No. 15)

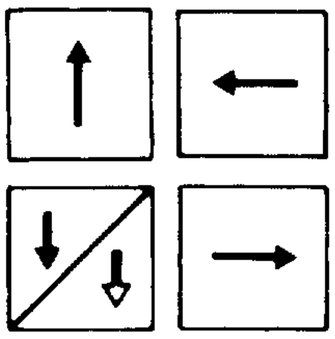
The single words that are to be changed are entered in the input line.
Only the numeric value of an address can be altered.

If the address is to be changed, the old word must first be deleted.
Only then can the new word be entered.

7.4.3 Deleting Single Words

In block N16, we want to delete the value X20.

Select the program as described in Section 7.2.



With the "Page" and "Cursor" keys (No. 22, 23, 24, 25) the cursor is moved in front of the word to be deleted (here X20).

X

Enter address X and terminate with the "Cancel" key (No.14).

The programmed X value has been deleted.

LF cannot be deleted. (It can only be deleted by deleting the entire block, as shown in Section 7.4.4).

7.4.4 Deleting Blocks

We want to delete block N16.

Select the program as described in Section 7.2 for block N16.

N16

Enter the block number and press the "Cancel" key (No. 14).

Block N16 is deleted.

NO

Blocks without a block number can be deleted by entering NO and by pressing the "Cancel" key (No. 14).

In the MDI-PP mode, when modes TE, TO or ZO are selected, the appropriate displays appear, but inputs are not possible (see the Display list, Section 1.5).

8. Manual Input of Zero Offsets, Tool Offsets, Test Data

Keyswitch (No. 38) enabled for input,

dependent on machine datum

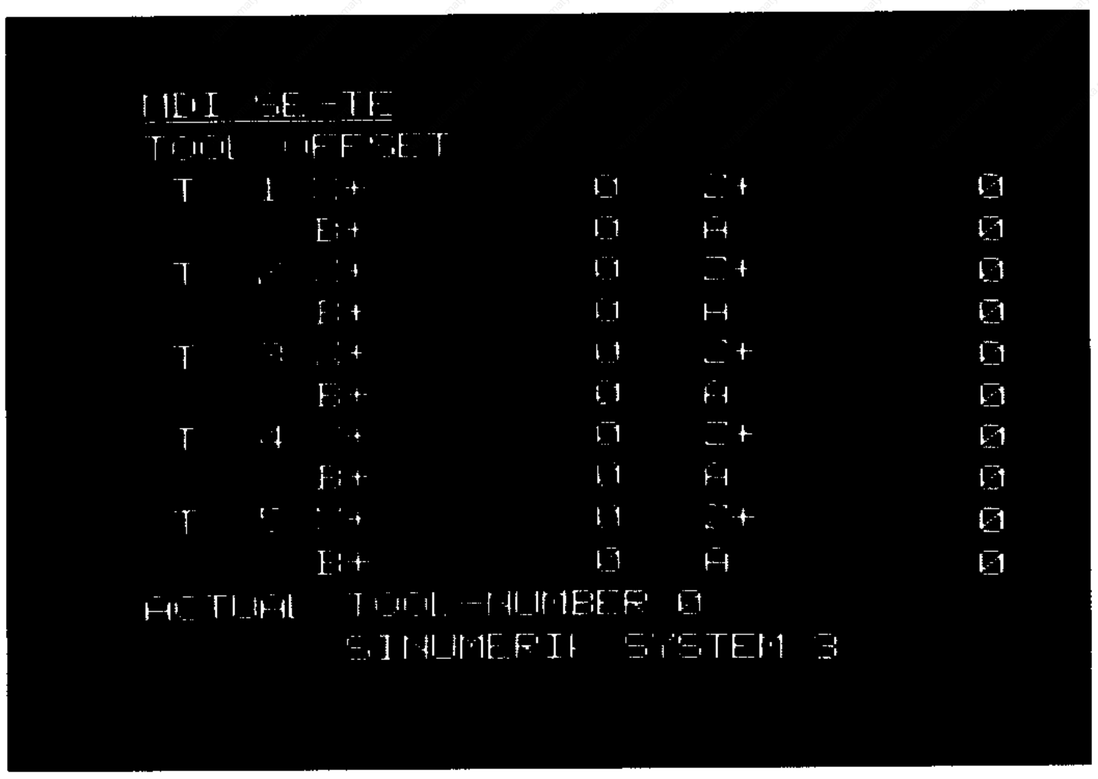
Operating mode MDI SE-TE (selector switch No. 31)

Selection:
With the identifiers (keys) "N" for testing or zero offset data or "D" (3M) or "T" (3T) for tool offset data and the desired parameter number and the "Page DOWN/Selection key", it is possible to select the ident numbers directly.
If an entry is missing or erroneous (faulty syntax, wrong iden-ti-fier, more than 3 decades), a page DOWN is executed. If the parameter number is higher than the maximum permitted (e.g. T100), the first page is selected. For axis-related machine data (100-273), the missing number can be selected, the result is not defined, however.

8.1 Tool Offsets

8.1.1 Tool Offsets 3 T

The tool offset mode is selected automatically (No. 6).

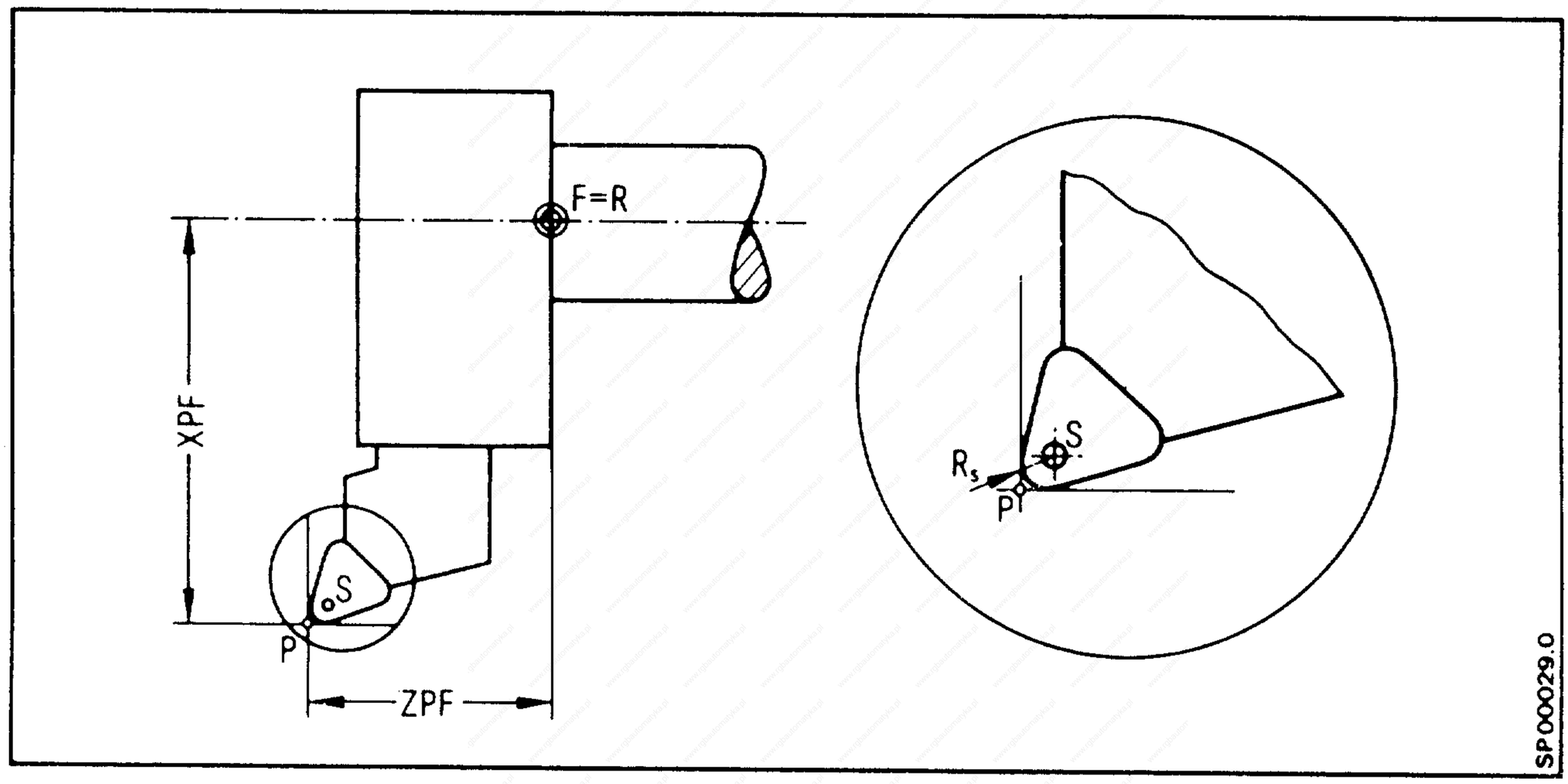


32 tool offset values can be selected. For lathes, 4 different values are necessary for an exact tool offset definition.

T1-T32: X and Z - tool geometry
B - tool cutter radius
A - tool position

The tool offset number of the tool being used is shown in the line 13 of the display.

8.1.1.1 Tool Cutting Point Position



P = Theoretical tip point

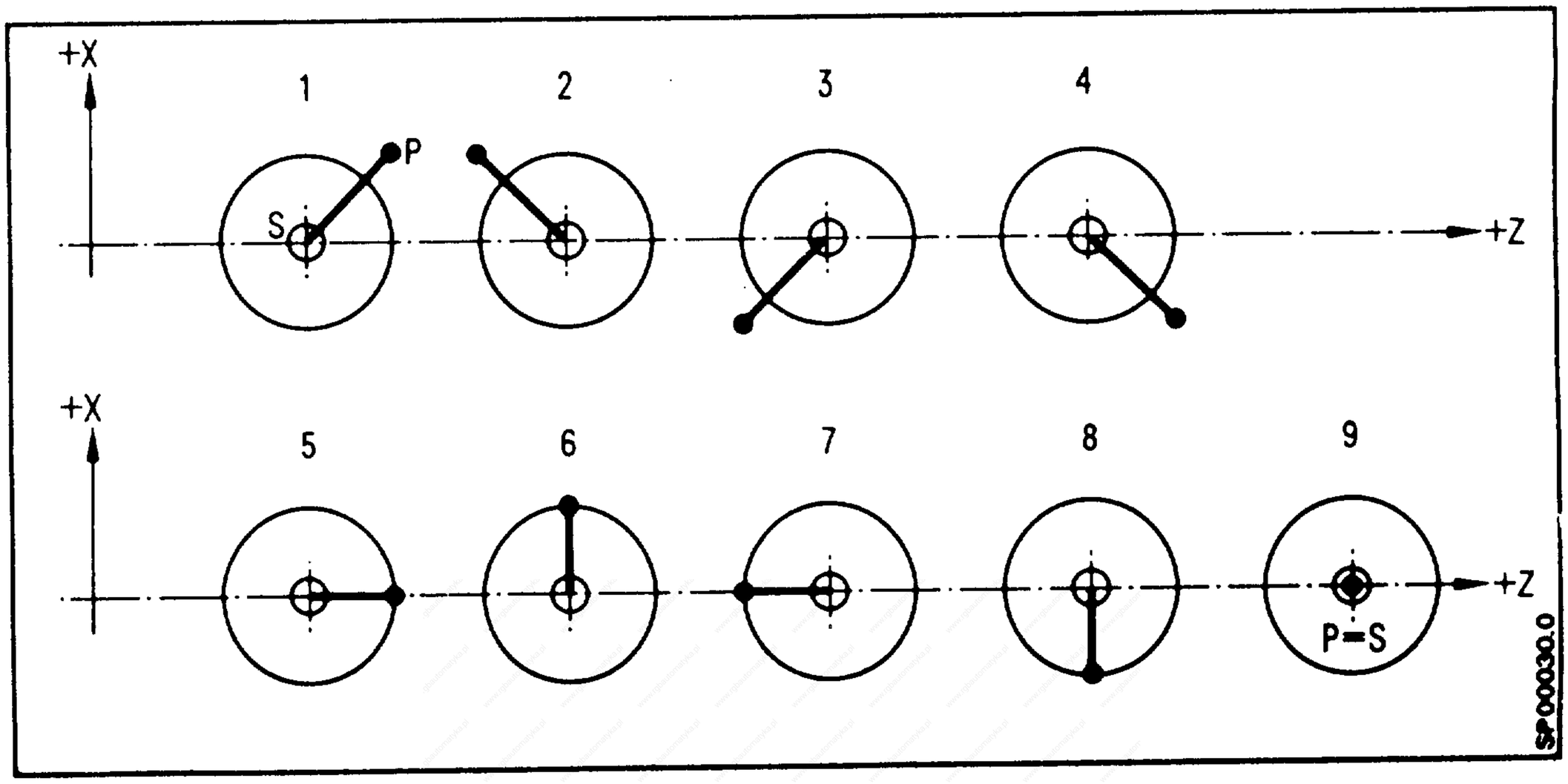
S = Tool tip radius center point

R_S = Tool tip radius

For the calculation of the equidistant points (left or right), the control needs four values, the fourth describing the position of the cutting edge.

The position of the tool tip "P" with respect to the tool tip radius center point "S" is defined by a total of nine codes.

The view is always from S to P.



8.1.2 Tool Offsets 3 M

Tool offset (No. 6) is selected automatically

```
MDI SE-TE
TOOL OFFSET
  D  1 D+          0 P+          0
  D  2 D+          0 P+          0
  D  3 D+          0 P+          0
  D  4 D+          0 P+          0
  D  5 D+          0 P+          0
  D  6 D+          0 P+          0
  D  7 D+          0 P+          0
  D  8 D+          0 P+          0
  D  9 D+          0 P+          0
  D 10 D+          0 P+          0
ACTUAL TOOL-NUMBER    0
        SINUMERIK SYSTEM 3
```

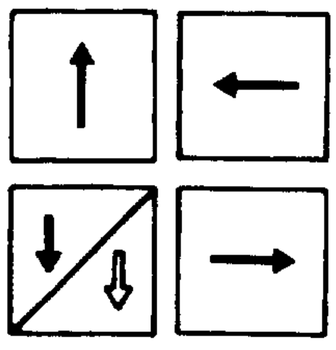
64 tool offset values an be selected.

D1-D64: D - cutter length

P - cutter radius

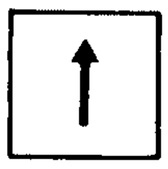
The active tool offset number is displayed in line 13.

8.1.3. Selecting a Particular Tool Offset Value

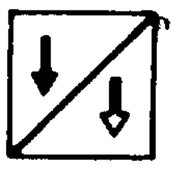


Preselect the desired tool offset by using the "Page" and "Cursor" keys (No. 22, 23, 24, 25).

It is also possible to make a selection by entering the key or "D" number (address, numeric keys) and pressing the page key.

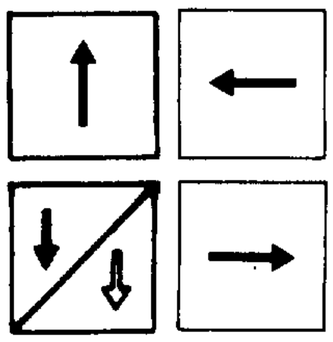


Return to the first offset value.



Continuous forward paging through offset value displays.

8.1.4 Offset Values, Input / Editing

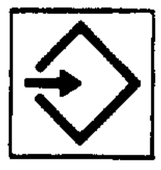


Preselect the desired tool offset number.

X Z B A Address keys (No. 11) for the 3 T

D P Address keys (No. 11) for the 3 M

Numbers (desired value)



Input key (No. 16). Store the value (acts as absolute value).

8.1.5 Tool Wear Edit

Edit key (No. 15) (acts additively)
Serves to enter tool wear within the range: ± 0.999 mm
± 0.0999 inch

The input range is checked during data entry. Input interlock using the keyswitch (No. 39) depends on an additional machine datum.
Tool wear editing is also possible in automatic operating mode (even when the program is running).

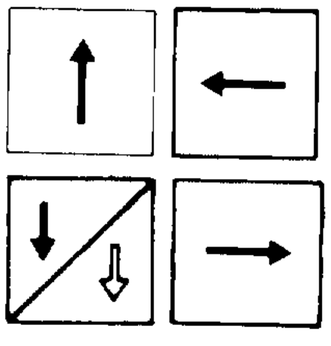
8.1.6 Deleting Offset Values

D - For the 3 M

T - For the 3 T

Delete all tool offsets (No. 14)

8.1.6.1 Deleting a Particular Offset Value



Preselect the desired tool offset number.

X Z B A Address keys (No. 11) for the 3 T

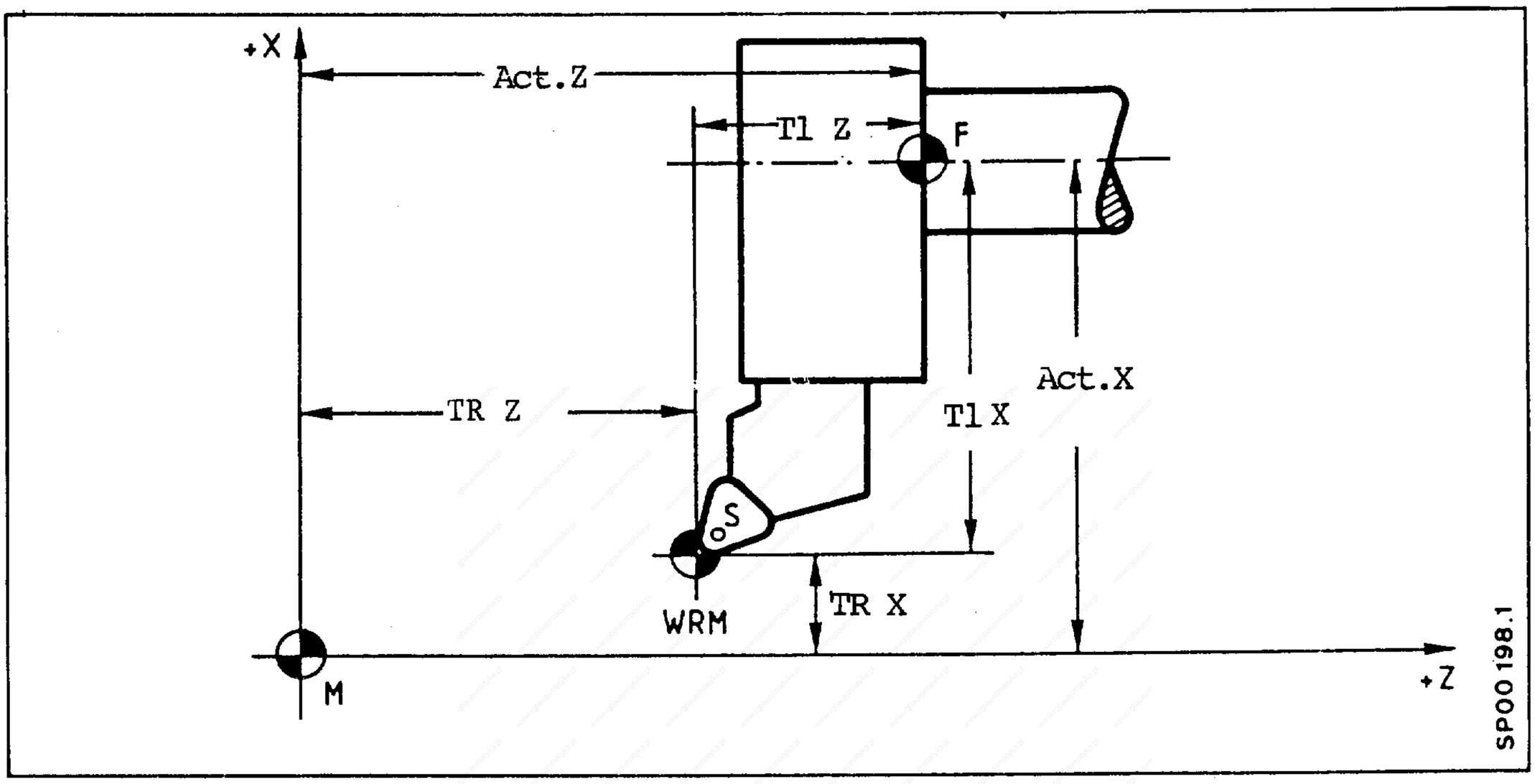
D P Address keys (No. 11) for the] M

"Cancel" key (No. 14)

The value selected with the address key is deleted.

8.1.7 Automatic Calculation of Tool Geometry data with the 3T

8.1.7.1 Determination of the Tool Length Compensation with Measuring Equipment (Crosshair) for the 3T



Act. X, Act. Z = actual coordinate of each axis

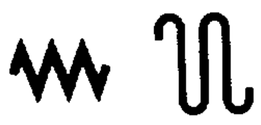
M = machine zero point

F = slide reference point

WRM = measuring device reference point (machine datum)

"Reference Point Approach" operating mode
(selector switch No. 31), see Section 10

The actual value memories are set to the reference value.



Operating mode JOG (selector switch No. 31)

Traverse to tool reference plane (TR) with the tool tip (set-up), e.g. by means of an optical system.

and/or

Operating mode INC Feed (selector switch No. 31)

Traverse to tool reference plane (TR) with the tool tip (set-up).

Select the tool offset mode (No. 6) and switch the display with display key (No. 17).

The following text is displayed in line 15:

```
TOOL-NUMBER CORRECT ?
```

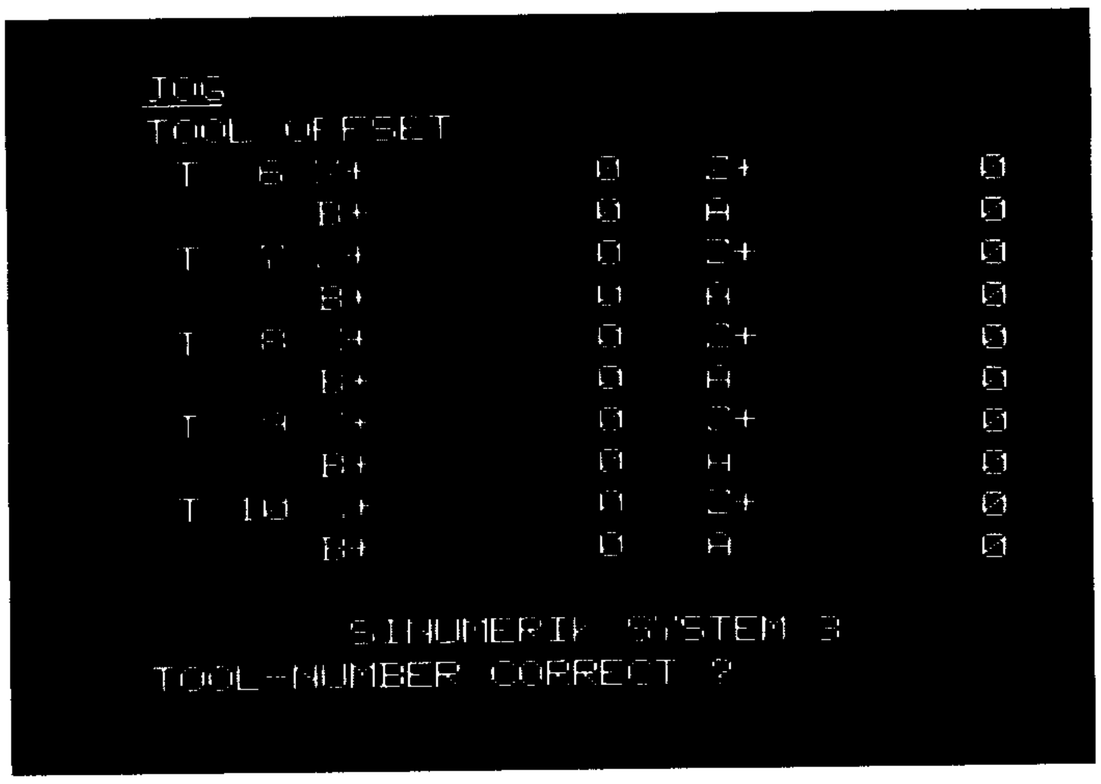
NO

Page through the tool offset display until the cursor is in front of the desired tool offset number.

YES

The selected tool offset number is correct.

The following display appears:



The "Calculation of Tool Length" automatic routine is initiated by pressing the "Start" key (No. 28)

Tool length = actual coordinate -tool reference value

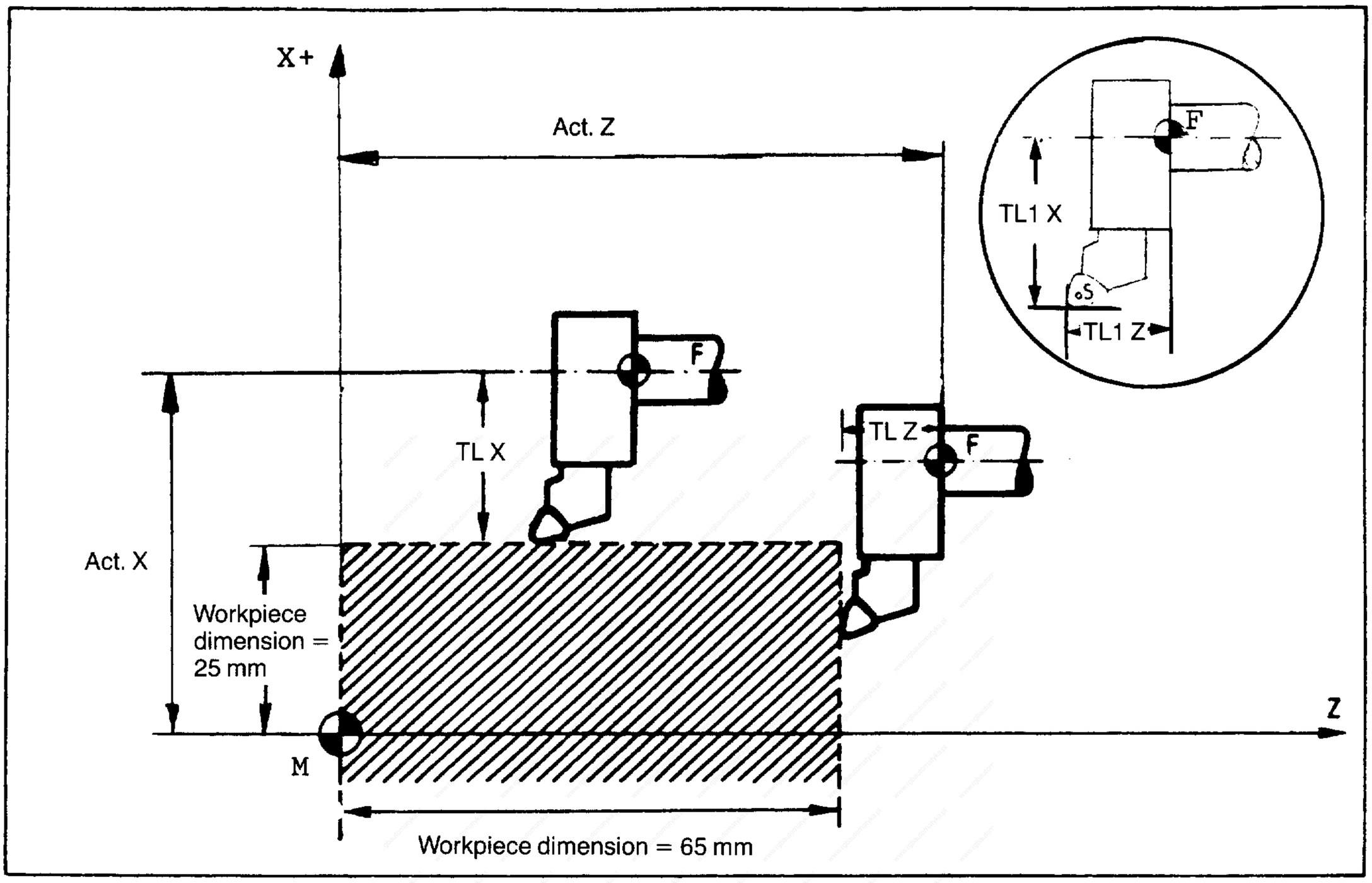
TL X = actual X - TR X

TL Z = actual Z - TR Z

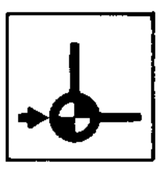
The tool lengths thus calculated for X and Z are stored in the preselected tool offset memory (here T8),

For the input of cutter radius and tool position, see Section 8.1.1.

8.1.7.2 Axis-by-Axis Determination of the Tool Length Compensation for the 3 T

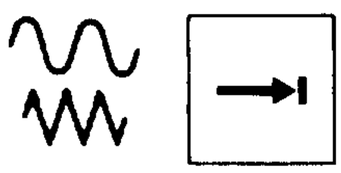


M = machine zero point
F = slide reference point
R = machine reference point
S = cutter radius centre

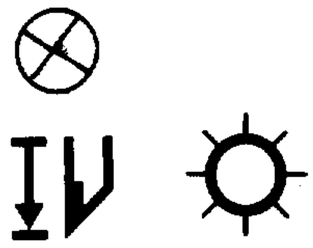


Reference point approach (Section 10)
Actual value memories are set to the reference dimension.

and/or



Select JOG and/or INC mode.
Approach a known workpiece point with the tool tip.



Select tool compensation with key (No. 17).
The following text appears in line 15:
Tool compensation no. correct?

with

NO YES Select the required tool compensation number.

and

The following text appears in line 15:

AXIS and VALUE

Z 65

Input of the workpiece value (e.g. Z65 mm)

of the tool tip

The following text appears in line 15:

START

The tool compensation value is computed by pressing the start key (No. 28).

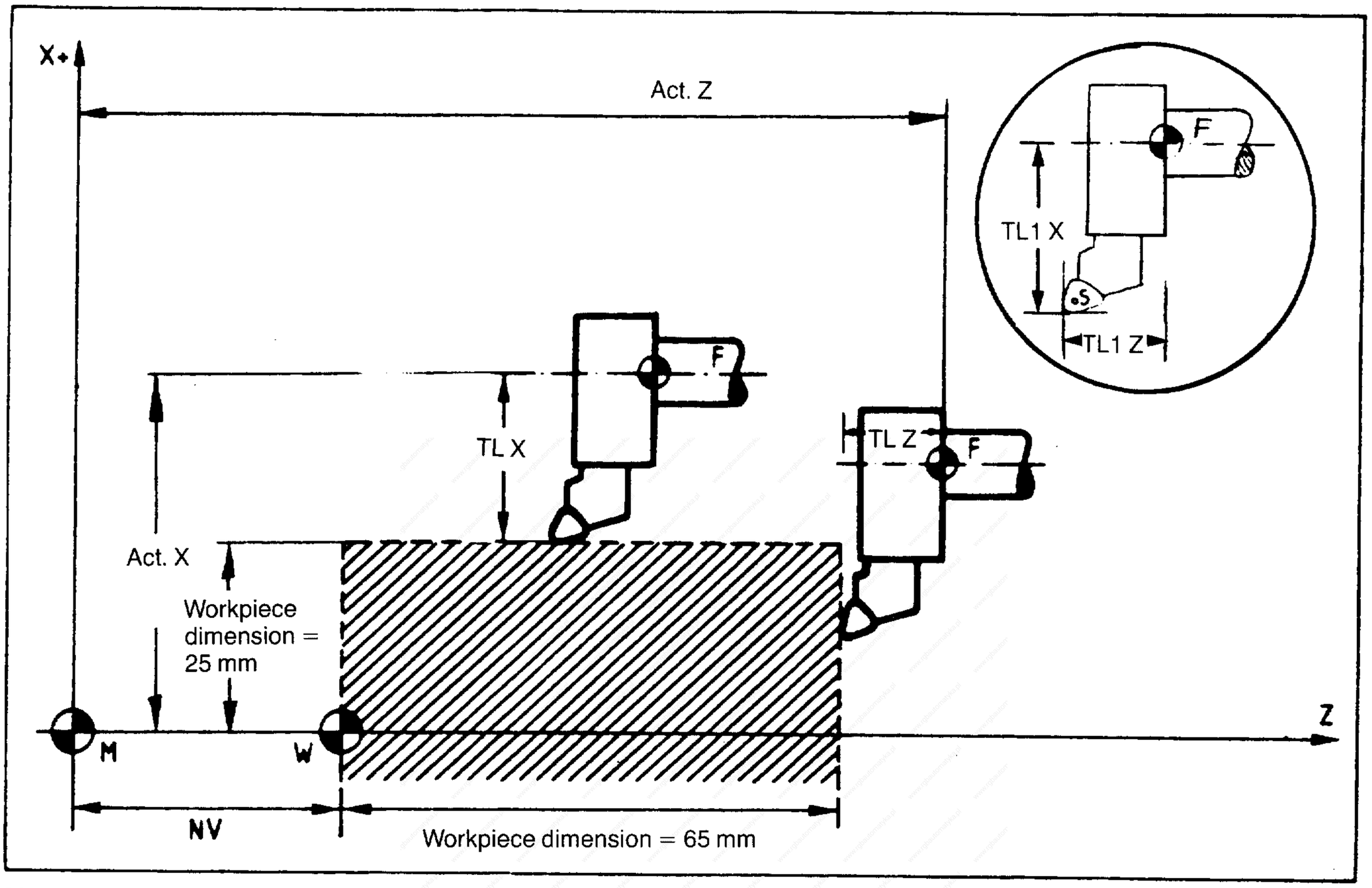
Tool length = actual value - workpiece dimension

T1 X = Actual X - Workpiece dimension X

T1 Z = Actual Z - Workpiece dimension Z

The tool length calculated in X or Z is stored in the selected tool compensation memory. The calculation has to be made for each axis separately. The axis input for the workpiece dimension serves as reference axis.

8.1.7.3 Axis-by-Axis Determination of the Tool Length Compensation taking the Zero Offset into account for the 3T



M = machine zero point R = machine reference point

F = slide reference point S = cutter radius centre

Reference point approach (Section 10)

Actual value memories are set to the reference value.

and/or

Select JOG and/or INC mode.

Approach a known workpiece point with the tool tip.

Select tool offset with key (No. 17).

The following text appears in line 15:

Tool compensation no. correct?

NO YES Select the required tool-offset number.

The following text appears in line 15:

AXIS AND VALUE

Z65

Input the workpiece dimension (e.g. Z65 mm) for the tool tip

The following text appears in line 15:

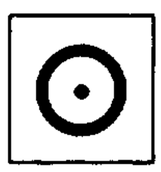
ZERO OFFSET NO.

Z1

Input the axis and zero offset number (1 to 12) which must be taken into account. The axis of the zero offset must coincide with the axis of the tool to be measured.

The following text appears in line 15:

START



The tool offset value is calculated by pressing the start key (No. 28).

Tool length = Actual value - (workpiece dimension + zero offset)

TL X = Actual X - (workpiece dimension X + zero offset X)

TL Z = Actual Z - (workpiece dimension Z + zero offset Z)

The tool length calculated in X or Z is stored in the selected tool offset memory, taking into account any zero offset. The calculation has to be made for each axis separately. The axis input for the workpiece dimension serves as axis reference

8.1.7.4 Determination of the Tool Length Compensation by means of Flying Measurement in JOG Mode for the 3T

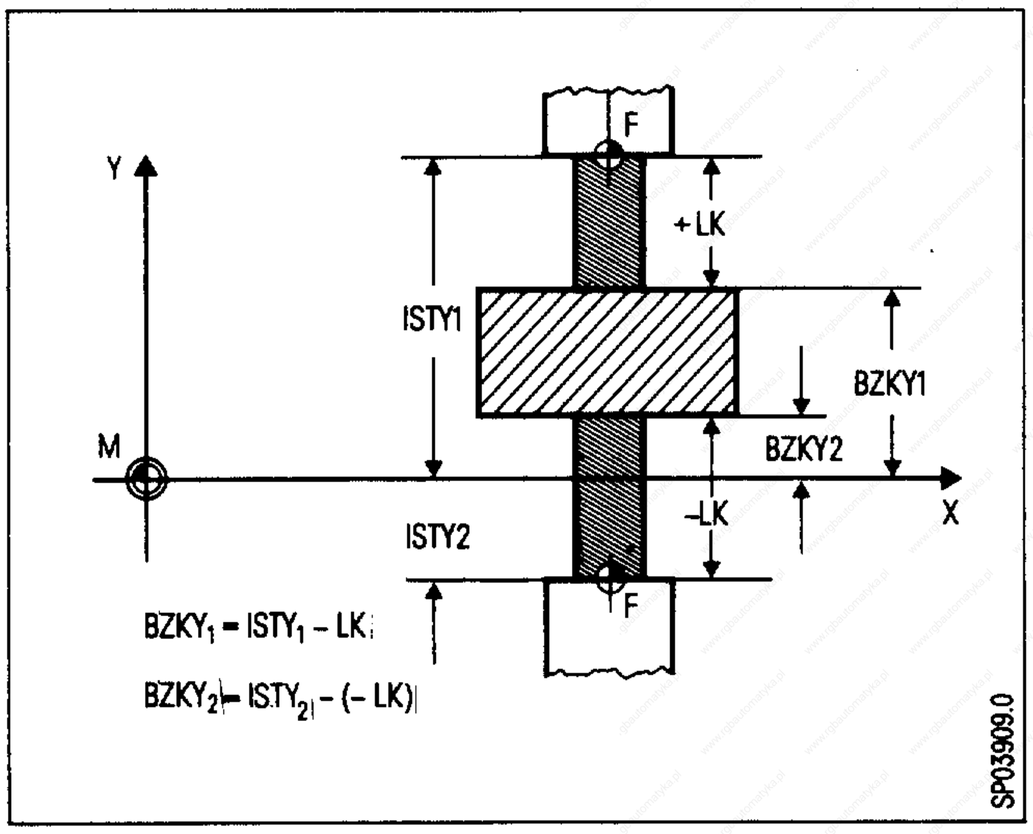
See Measuring Cycle Description, Part 1.

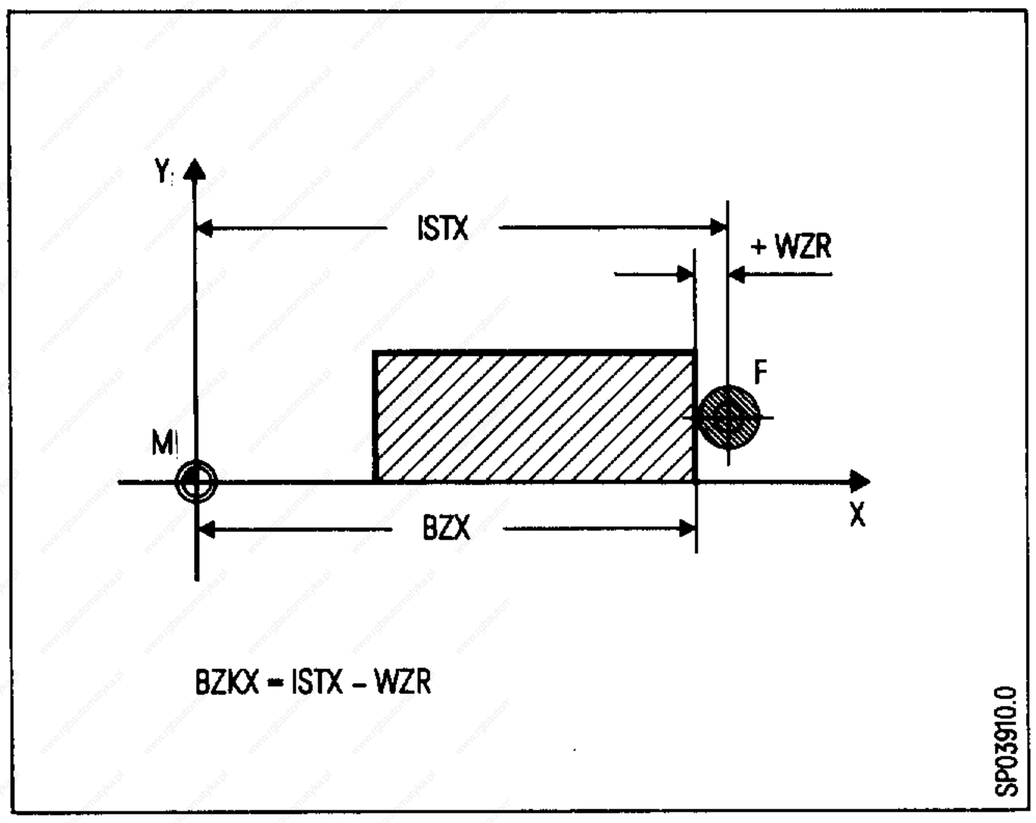
8.1.8 Automatic Determination of Tool Offset for the 3M

Before the tool offset can be calculated, the reference edge for the axis in question must be determined. For this, a reference tool with known dimensions is used. If a tool is to be measured in an axis in which the reference edge has not been determined, an alarm is triggered. Reference edge values are returned even after the NC is switched off.

8.1.8.1 Calculating the Reference Edge

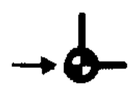
Operating sequence using operator prompting





M = Machine zero point

F = Tool reference point



For reference point approach mode (selector switch No. 31)
See Section 10.
Actual value memories are set to the reference dimension.

Select JOG mode (selector switch No. 31) and approach a workpiece surface with a reference tool.

and/or

Select INC mode (selector switch No. 31) and approach a workpiece surface with a reference tool.

Select Tool Offset mode (No. 7) using the Display Switch-over Key (No. 17).

The following text is displayed in line 15:
TOOL OFFSET NO. CORRECT?

with

Position cursor to desired tool offset memory.

and

Selected tool offset memory is correct

The following text is displayed:
ENTER REFERENCE TOOL?

because reference edge must be determined

The following text is displayed:
TOOL DIMENSION

Input tool dimension (radius or length) with sign

The following text appears:
AXIS

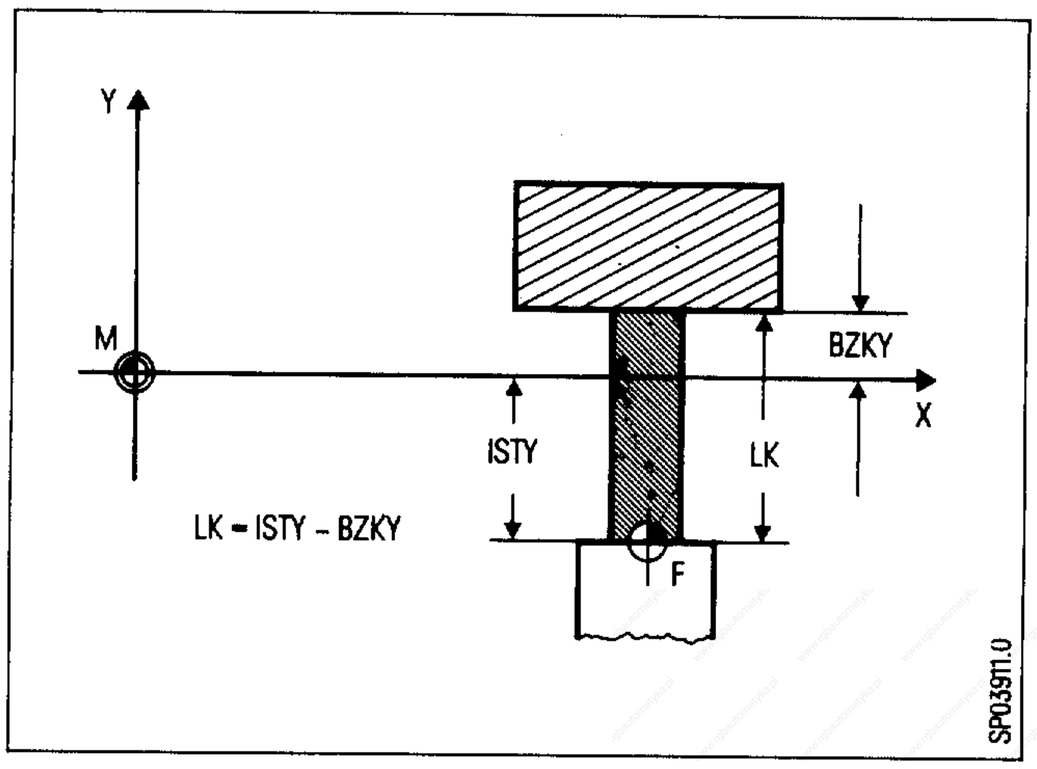
Enter address name of the reference edge

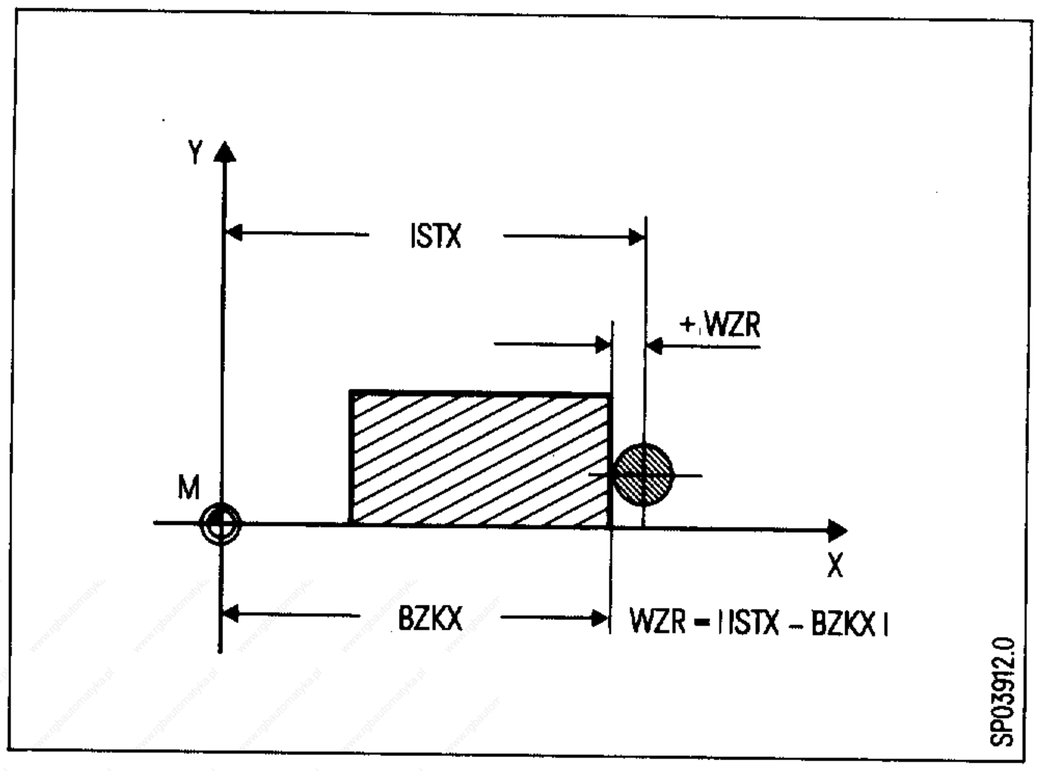
The following text appears:
START

Calculate reference edge and store in internal memory.

8.1.8.2 Calculating the Tool Offset

Operating sequence with operator prompting





M = Machine zero point

F = Tool reference point

For reference point approach mode (selector switch No. 31), See Section 10

Actual value memories are set to reference dimension

Select JOG mode (selector switch No. 31) and approach reference edge with tool.

and/or

Select INC mode (selector switch 31) and approach with tool.

Select tool offset mode (No. 7) using the Display Switchover key (No. 17)

The following text is displayed in line 15:
TOOL OFFSET NO. CORRECT?

with **NO** Position cursor to desired tool offset memory.

and **YES** the selected tool offset is correct.
The following text is displayed:
ENTER REFERENCE TOOL?

NO because tool dimension must be determined.

The following text is displayed:
CALCULATE TOOL RADIUS?

YES "Continue with axis"

NO The following text is displayed:
CALCULATE TOOL LENGTH?

NO "Calculate jump-back to tool radius"

YES The following text is displayed:
AXIS

 Enter name of axis approached

The following text is displayed:
START

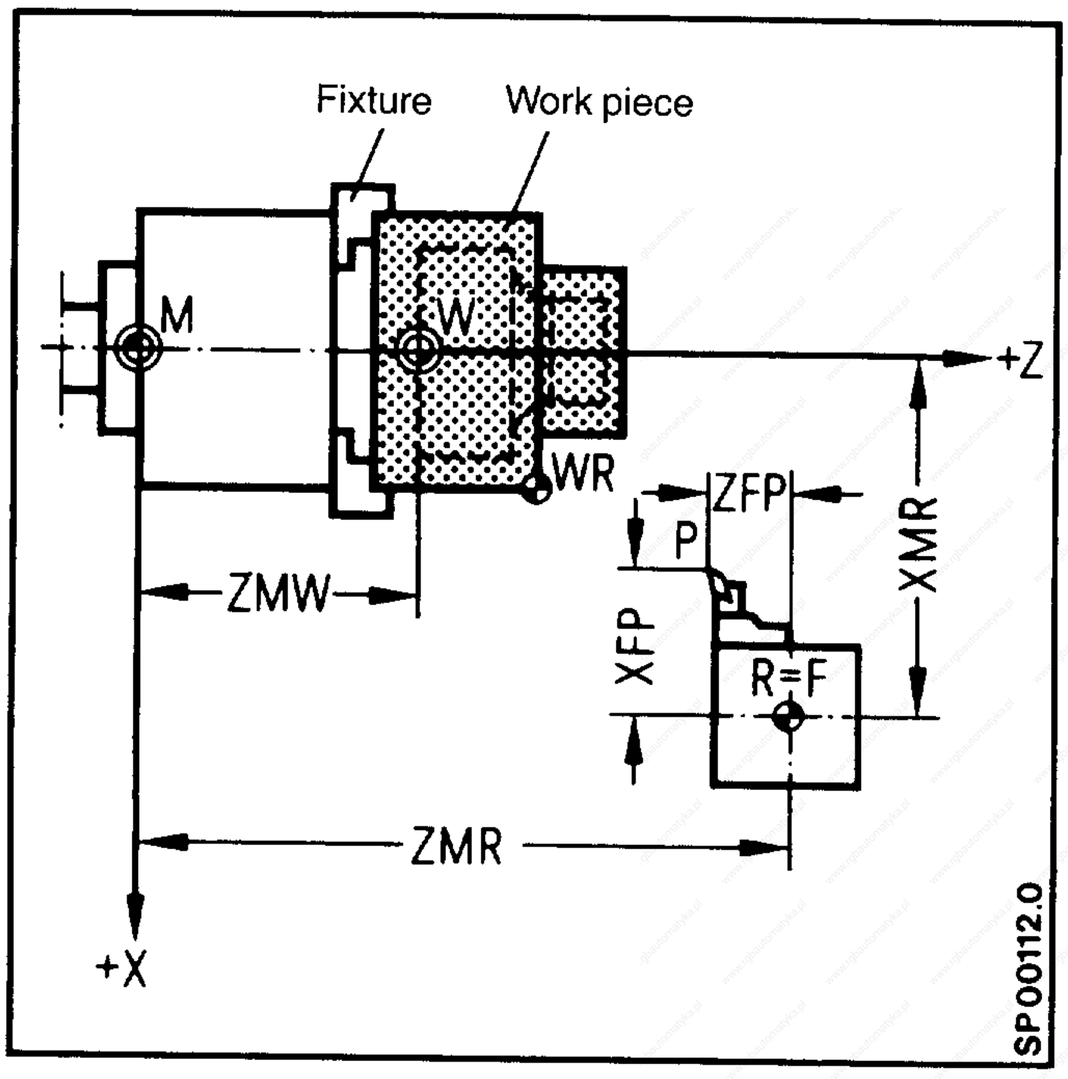
 Calculate tool offset (length or radius) and store in the selected offset memory.

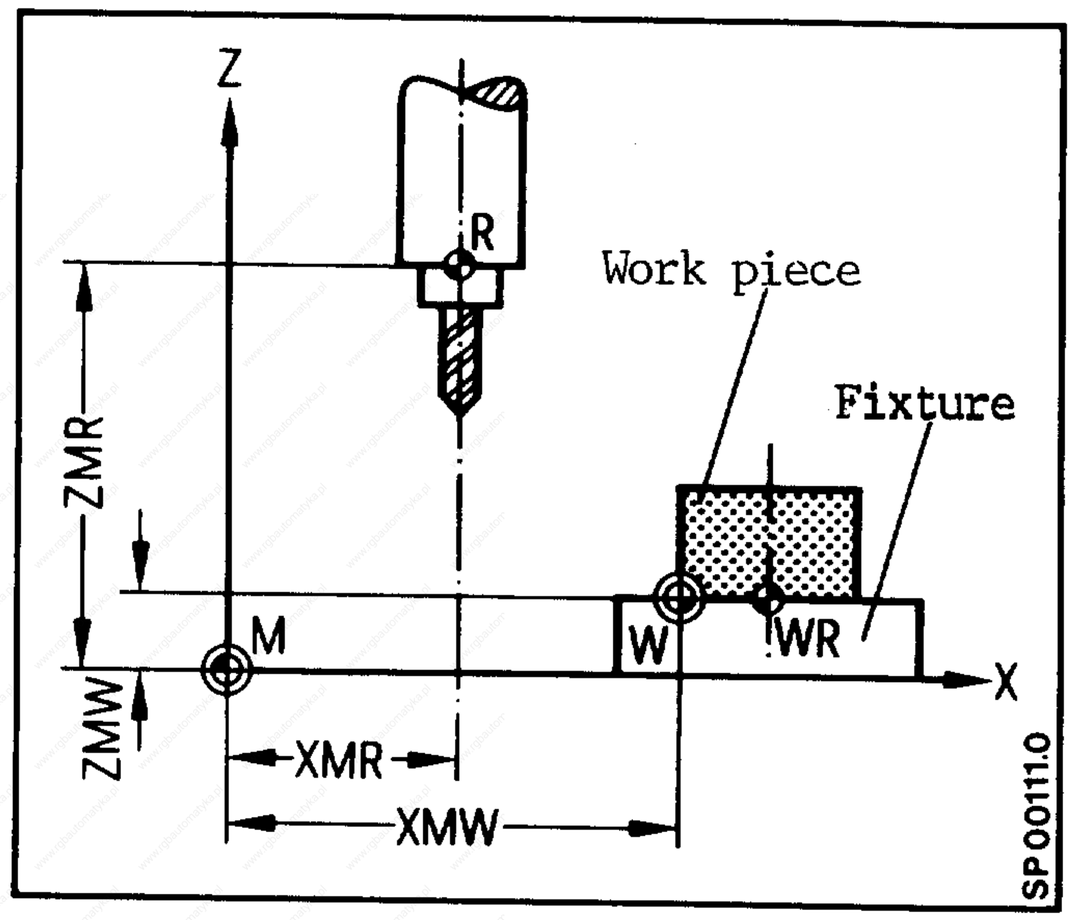
This can than be implemented for any tool (length/radius).

8.2 Zero Offsets

8.2.1 Reference Points

The appropriate reference point coordinate values are entered automatically into the actual value memory counter after the return to the reference point. The reference point coordinates are the fixed distances between the reference point R and the machine zero point M.





Reference point diagram

F	= slide reference point
M	= machine zero point
W	= workpiece zero point
R	= machine reference point
WR	= workpiece reference point
ZMR, XMR, etc.	= reference point coordinate for each axis
XMW, ZMW, etc.	= the sum of all zero offsets for each axis

ZO = preset ZO (G54...) + add. ZO (G59 + ext. ZO (PLC)

8.2.2 Selectable Zero Offset (XMW, ...)

After a reference prkpiece zero point

R = machine reference point

WR = workpiece reference point

ZMR, XMR, etc. = reference point coordinate for each axis

XMW, ZMW, etc. = the sum of all zero offsets for each axis

| ZO = preset ZO (G54...) + add. ZO (G59 + ext. ZO (PLC) |

After a reference point approach, the actual value memory and the position read-out are referenced to the machine zero point M.

The machining program for the workpiece is referenced to the workpiece zero.

The machine zero point M and the workpiece zero point W do not have to coincide. Depending on the nature and the clamping of the workpiece, the distance between the machine zero point M and the workpiece zero point W may vary.

A settable zero offset (ZO) is defined and entered so that this variable XMW distance need not be taken into account in the program. In the program blocks, the settable zero offset distance is included in the axis commands each time that axis is programmed.

WARNING: Zero offsets are always taken into account (in absolute and incremental programming). When a program is run in incremental dimensions the operator must make sure that the settable zero offset has been cancelled.

8.2.3 Measuring the Zero Offset

Upon reference point approach, the actual value memory is referenced to the machine zero point M.

After clamping the workpiece, the tool traverses in jog mode to the workpiece zero point W or to the workpiece reference point WR.

When the workpiece zero point W is reached, the settable zero offset can be calculated as follows:

Settable ZO = Displayed position - Tool length

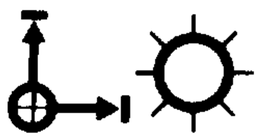
e.g. | XMW = XMF - XFP |

If the workpiece reference point was approached the formula is:

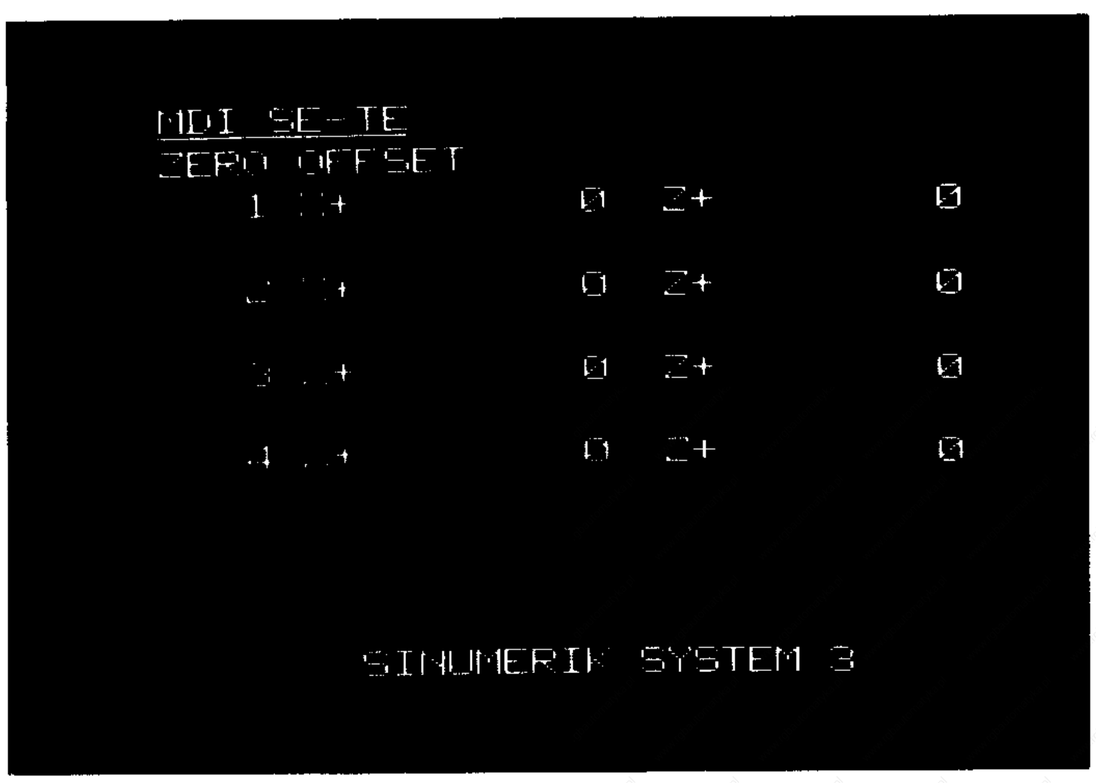
Settable ZO = Displayed Position - tool length - difference (tool reference point/workpiece zero point

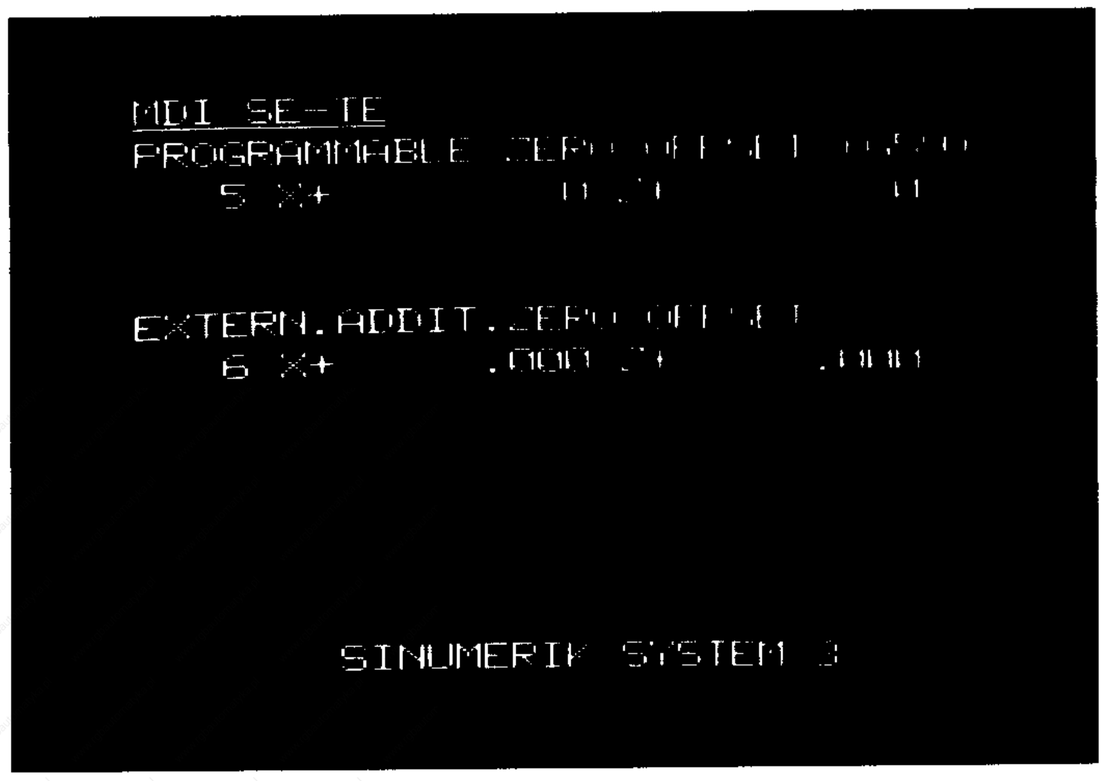
e.g. | XMW = XMF - XFP - WR |

8.2.4 Zero Offset Display

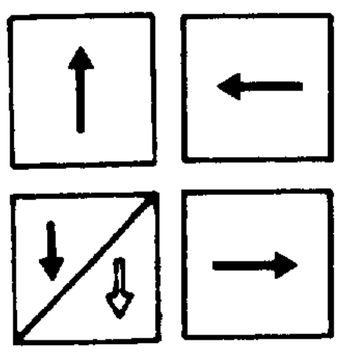


Select Zero Offset mode (No. 7) using the Display changeover key (No. 17)





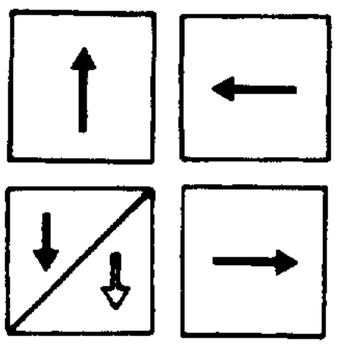
Display of:
-settable zero offset (1-max. 12)
-programmable zero offset
(including the coordinate turning angle)
-external zero offset
for all the axes selected.



Preselect the desired zero offset

No inputs are possible with the programmable (5) and external (6) zero offsets.

8.2.5 Deleting Settable Zero Offsets



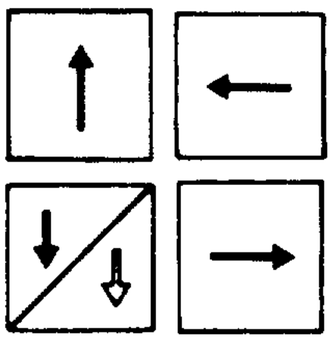
Select the desired number with the "Page" and "Cursor" keys (No. 22, 23, 24, 25).

X Z Address keys (No. 11) for the 3 T

X Y Z 4 Address keys (No. 11) for the 3M

The value is deleted with "Cancel" (No. 14) and the corresponding address keys.

8.2.6 Entering Settable Zero Offsets

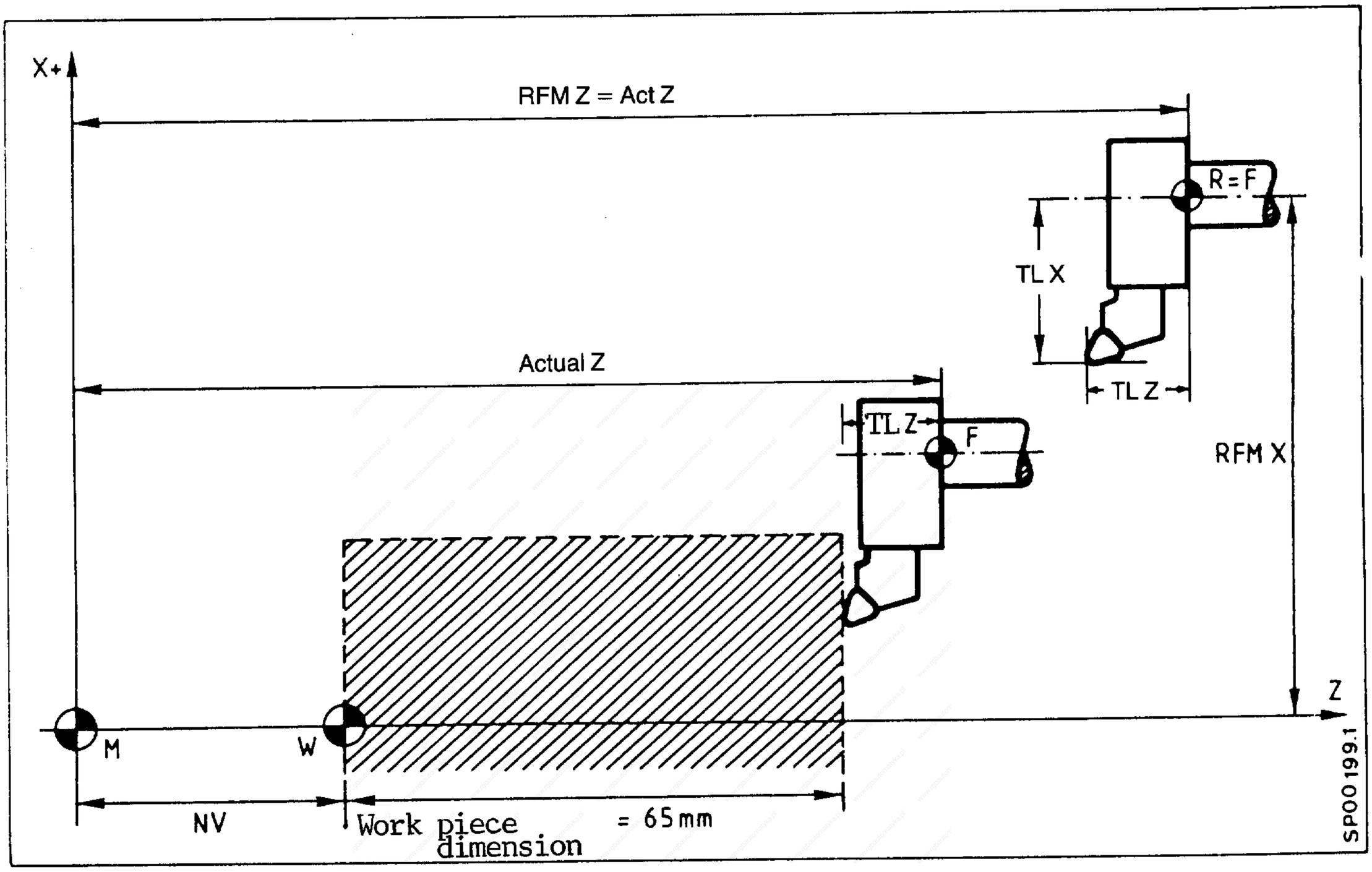


Preselect the desired zero offset

e.g. X1000. The new value is stored for the X axis by inserting the X-value and then pressing the input key (No. 16).

8.2.7 Automatic Determination of the Zero Offset for the 3 T

Operation sequence using operator prompting (guidance)



M = machine zero point

F = slide reference point

R = machine reference point

W = workpiece zero point

For Reference Point Approach mode
(selector switch No. 31) see Section 10
Actual value counters are set to reference dimension.

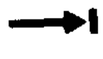
JOG mode (selector switch No. 31)

Traverse the tool tip to a known point on the workpiece

(set-up)

(Use the workpiece drawing.)

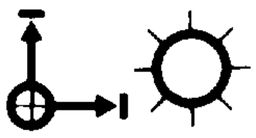
and/or



INC mode (selector switch No. 31)

Traverse the tool tip to a known point on the workpiece

(set-up)



Select the Zero Offset mode (No. 7) with the Display Change-over Switch (No. 17)

Line 15 displays the following text.

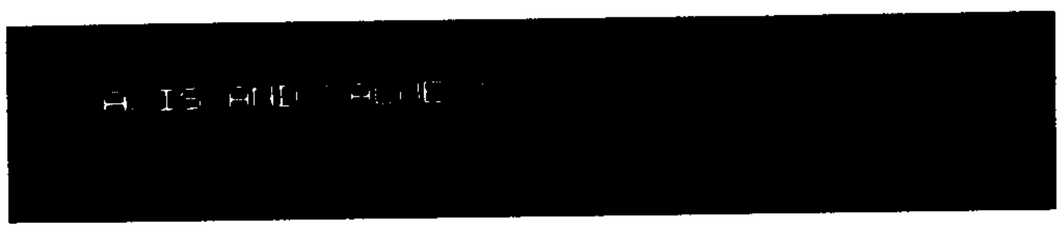
NO

Page through the Zero Offset display until the cursor is in front of the desired zero offset number.

YES

Zero offset number displayed is correct.

The following text is displayed in line 15:



X 65 — Enter the workpiece dimension associated with the tool tip (e.g. X + 65 mm).

The following text is displayed in line 15:

T12 — Enter the tool offset number for the tool being used on workpiece.

The following text is displayed in line 15:

The "Calculation of Zero Offset" automatic routine is initiated by pressing the Start key (No. 28).

Zero offset = actual value -
(workpiece dimension + tool length)

ZO X = actual X - (WX + tool length X)

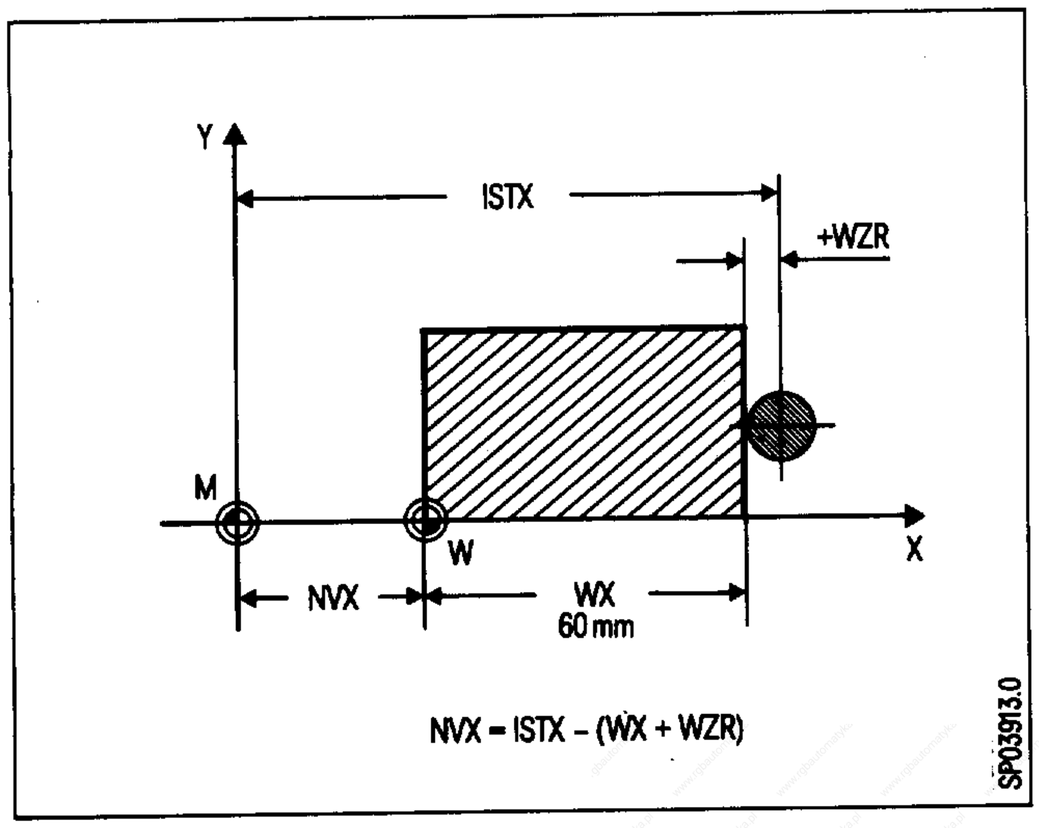
ZO Z = actual Z - (WZ + tool length Z)

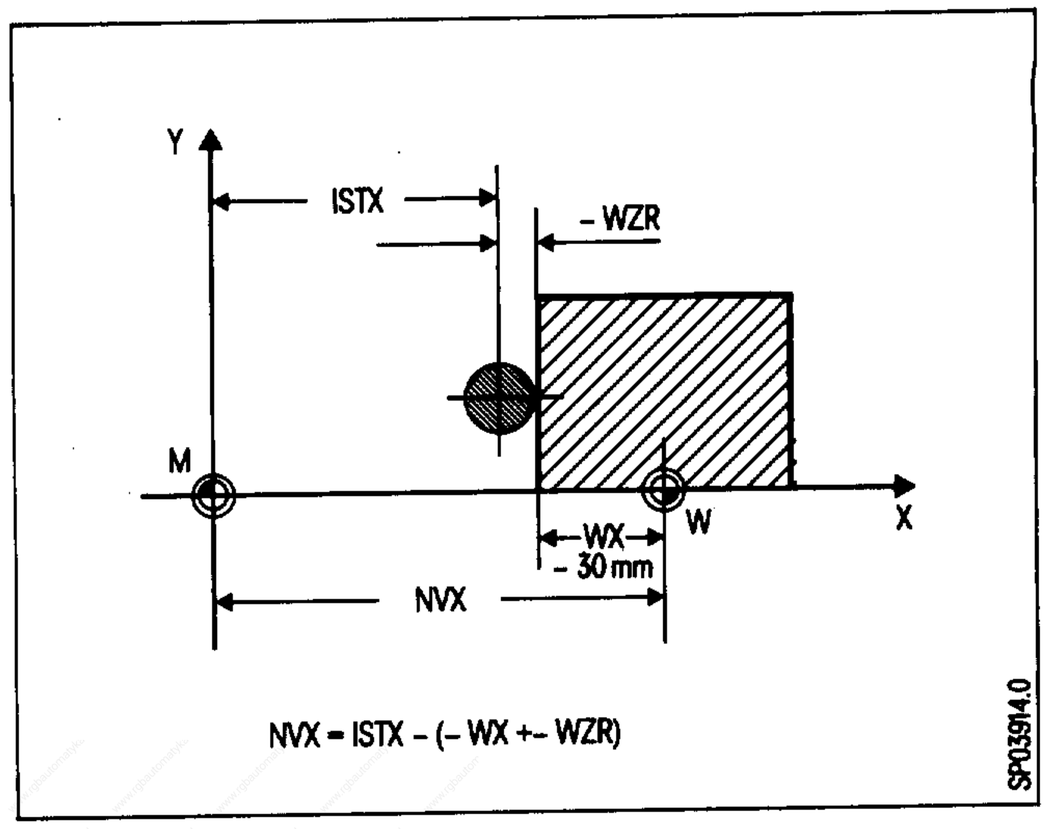
The zero offsets calculated for X and Z are stored in the preselected zero offset memory (here, e.g.1).

The automatic calculation of the zero offsets has to be carried out separately for each axis. The axis entered with the workpiece dimensions serves as the reference axis.

8.2.8 Automatic Determination of the Zero Offset for the 3M

Operating sequence with operator prompting (guidance)





M = Machine zero point

W = Tool zero point

For Reference Point Approach modes (selector switch No. 31), See Section 10.

Actual value memories are set to reference dimensions.

Select JOG mode (selector switch No. 31) and approach a workpiece surface with the tool.

and/or

Select INC mode (selector switch No. 31) and approach a workpiece surface with a tool.

Select zero offset mode (No. 7) using the display change-over key (No. 17).

The following text appears in line 15:
ZERO OFFSET NO. CORRECT?

with

NO

and

YES

Page through zero offset displays until the cursor is at the desired zero offset number

selected zero offset number is correct.

The following text is displayed:

AXIS AND VALUE

X60

(X-30)

Input of the workpiece dimension of the approached surface (e.g. 60 mm, - 30 mm)

The following text is displayed:

WORKPIECE DIMENSION

Input of workpiece dimension (radius or length) with sign

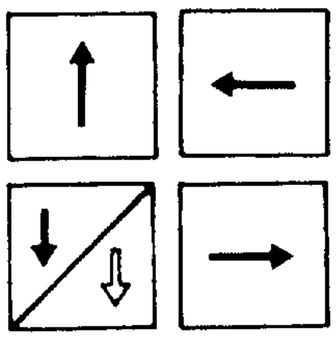
The following text is displayed:

START

The zero offset is calculated and stored in the preselected zero offset memory.
Automation determination of zero offset must be executed for each axis separately. The axis input for the workpiece dimension serves as reference axis.

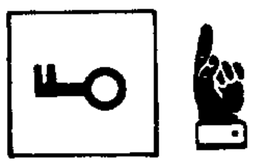
8.3 Programmable Spindle Speed Limitation

The speed limitation programmed under G92 S... (see the Programming Instructions), is displayed and can be modified by using the input keys.



Select display 20 by using the "Page" and "Cursor" keys (No. 22, 23, 24, 25).

```
MDI SE-TE
SPINDLE SPEED LIMITATION (G92)
> 20  S          0


ORIENTATED SPINDLE STOP  (M19)
  22  S         .0




          SINUMERIK SYSTEM 3
```



Input can be interlocked with the keyswitch (No. 38)

(dependent on machine datum)

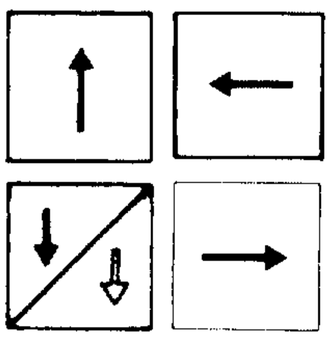
S 800

Enter the limit speed

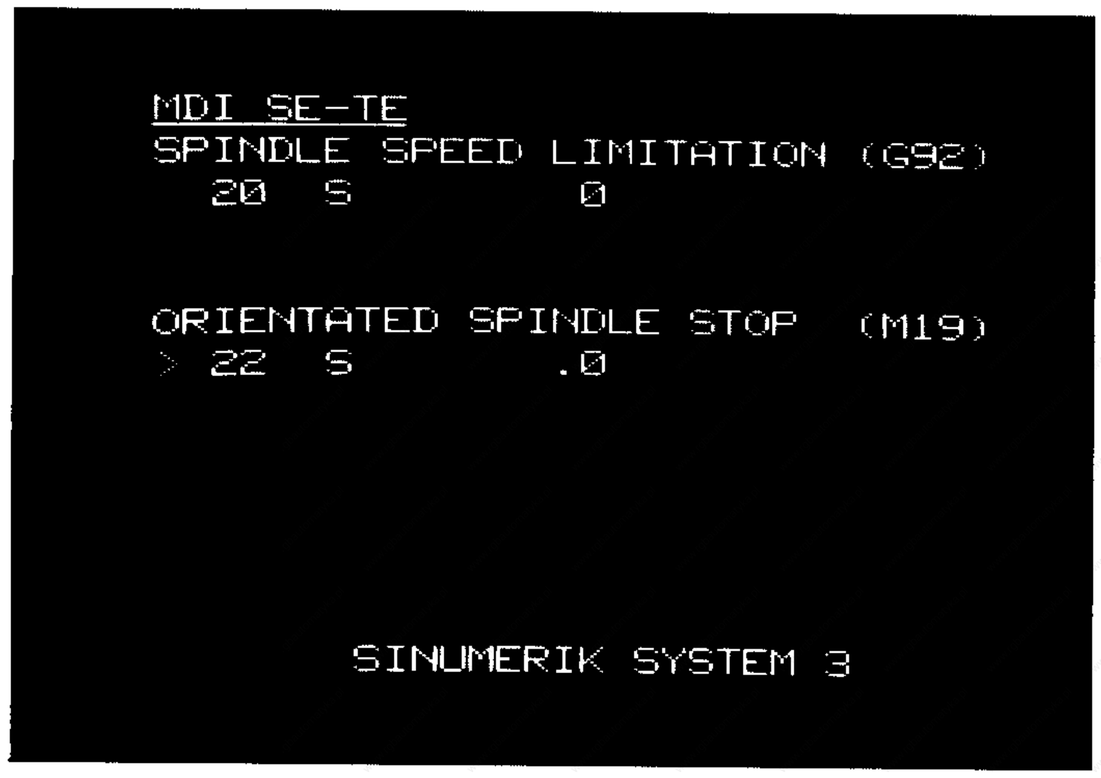
(e.g. 800 rev/min)

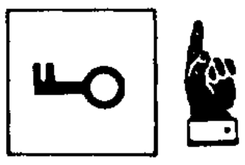
8.3.1 Oriented Spindle Stop

The oriented spindle stop programmed under M19 S... (see the Programming Instructions), is displayed and can be modified by using the input keys.



Select the display with the "Page" or "Cursor" keys (No. 22, 23, 24, 25)

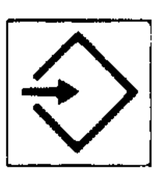




Input can be interlocked with keyswitch (No. 38)

(dependent on machine datum)

S 156.5

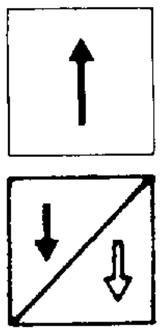


Input S is in degrees (here 156.5o) from the zero point, viewed in the direction of M03.

8.3.2 Absolute starting angle offset

The absolute starting angle offset for the level-up thread function is entered in the same way as program or spindle speed restriction or oriented spindle.

8.4 Bit-oriented User Data

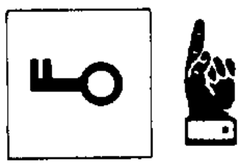


Select the display with the "Page" keys (No. 2, 23)

```
MDI SE-TE
SETTING DATA NUMBER 25
 N
 0
 0
 0
 0 CODE PUNCH OUT     0=ISO 1=EIA
 0
 0 PROGRAM START WITH LF     1=ON
 0 TAPE BLOCK PARITY 0=OFF  1=ON
 0 OPERATOR GUIDING  0=OFF  1=ON
 0 DISPLAY WORKPIECE SYSTM 1=ON
```

```
MDI SE-TE
SETTING DATA NUMBER 26
 N
 0
 0
 0
 0
 0
 0
 0
 0 HANDWHEEL CALIBRATION 100
 0 HANDWHEEL CALIBRATION 10
 0 HANDWHEEL CALIBRATION 1

        SINUMERIK SYSTEM 3

                N00000100
```

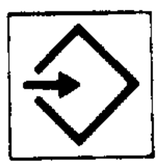
The user data are presented in plain text.



Input can be interlocked with keyswitch (No. 38)

(dependent on machine datum)

N11101010 Input user datum

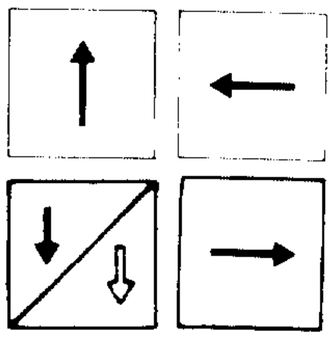


The Appendix contains a list of the bit-oriented user data.

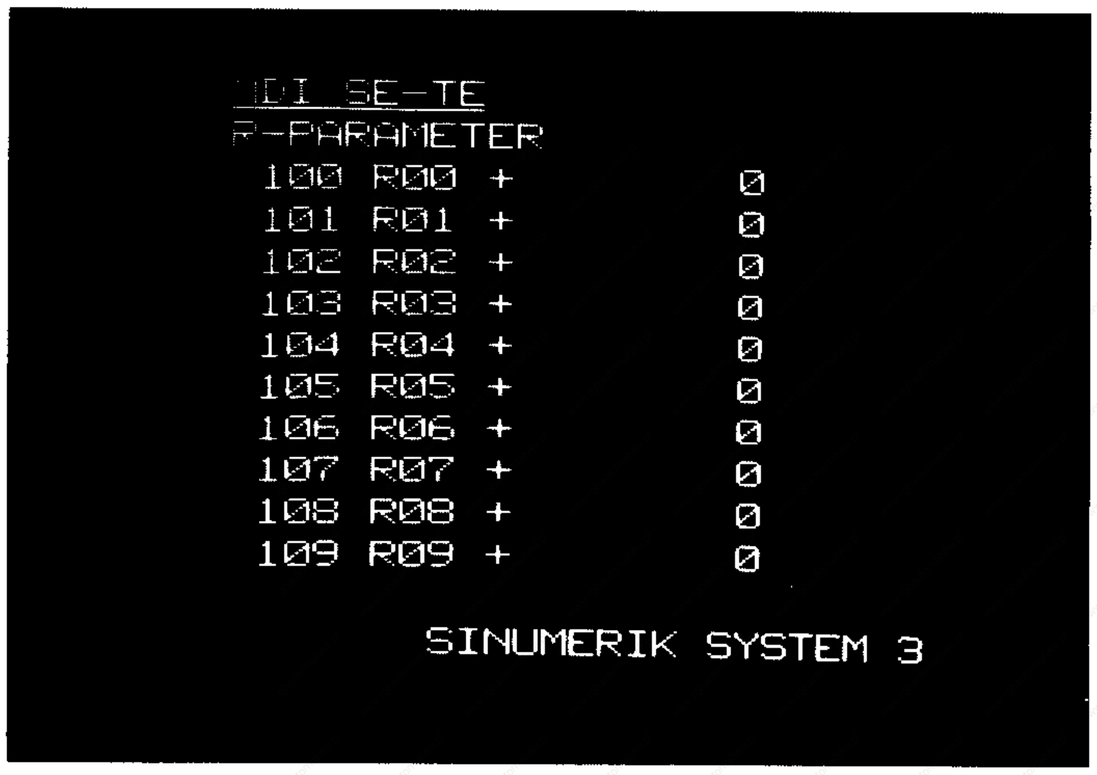
When entering the data, the leading zeros can be disregarded.

8.5 R parameters

The R parameters (see the Programming Instructions) can be displayed and modified.



Display numbers 100 to 599 (for groups 0 to 4: parameters R00 to R99) can be selected with page keys (No. 22, 23) and cursor keys (No. 24, 25). It is also possible to make a selection by entering N100 to N599 and then pressing the page key.



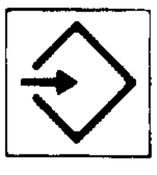
Keyswitch interlock (No. 38)

(dependent on machine datum)

R00
1000.5

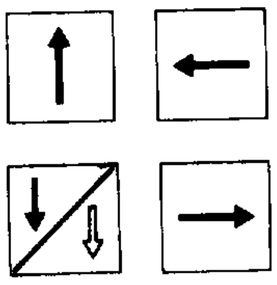
Entering an R parameter

e.g. R00 1000.5



8.5.1 Background memory

The background memory for calculation parameters can be displayed and modified.



Select display number 600 to 699 (parameters 00 to 99) using the "Page" keys (No. 22, 23) and the "cursor" keys (No. 24,25)

It is also possible to make a selection by entering N600 to N699 and then pressing the page key.

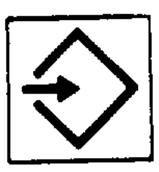
Machine data protection switch interlock
(on the logics component)

Keyswitch interlock (No. 38)
(dependent on machine datum)

Input limit interlocked via machine data

H00
1234.5

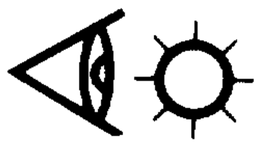
Input of a background parameter using the address letter H.
e.g. background memory 00 = 1234.5



The address letter H is used only for the input of values to the background memory via the operator panel. In the NC parts program the background memory can only be written into or read via @29.

8.6 Data and Selection Displays

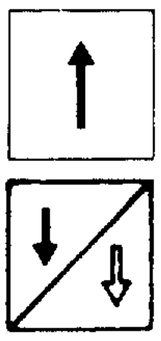
Select "Test" (No. 8) using appropriate key



Change the display (No. 17)

The following data are displayed under the TEST display:

Ident. Number	Data Type
1	Alarms NC
7 - 9	NC input signals
10 - 11	NC output signals
100 - 755	machine data
800 - 861	service values



Paging through the display numbers with page keys (No. 22, 23). It is also possible to input N1 to N861 and then press the page key.

Value can only be changed in the MDI SE-TE mode (selector switch No. 31), and are protected by using the machine data lock switch (see the Commissioning Instructions).

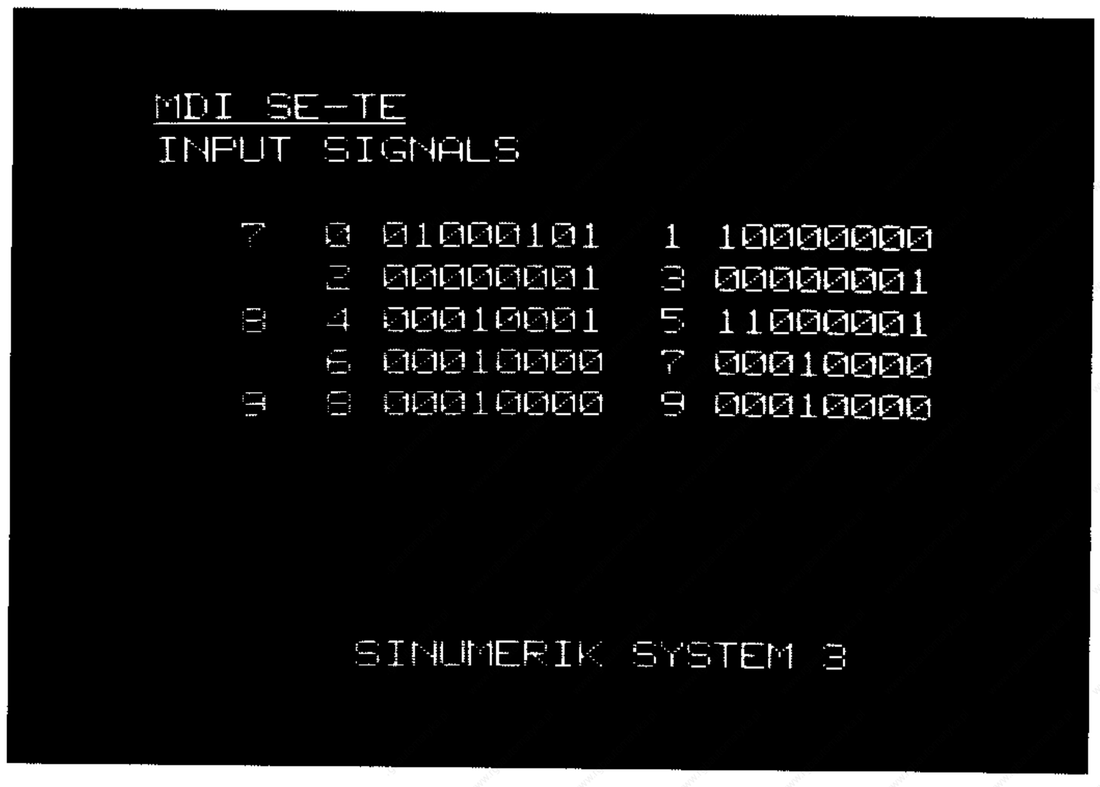
8.6.1 Alarm Display

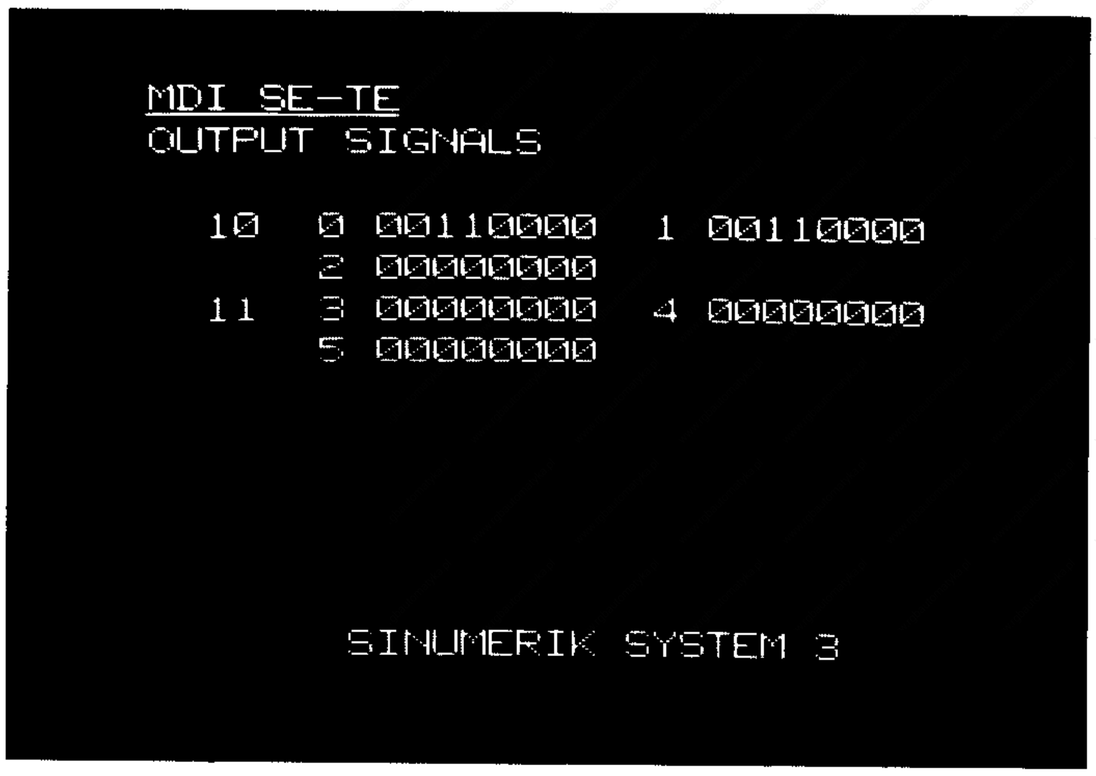
The first NC alarm number of the alarms present is generally shown in the 14th line of the display. Optionally, all alarms present can be displayed in ascending order at intervals of 2 seconds. In addition, a maximum of 10 simultaneous NC alarms are displayed in plaintext (brief description of error search).

8.6.2 Interface Image

8.6.2.1 NC/PLC input-output signals

The I/O signals of the interface between the NC and the PLC can be displayed at any time with the help of the integrated interface diagnostics. No inputs are possible. See the following tables for the individual interface signals and their selection.





SINUMERIK SYSTEM 3		INPUT INTERFACE 3M				3M4C			23.09.88
TESTING No.	Byte	Data bit number 7	6	5	4	3	2	1	0
7	0	Operating mode switch D	C	B	A	Feedrate/rapid override switch D	C	B	A
7	1	Key switch	Dry-run	Skip block	Single block	Block search	Spindle override switch C	B	A
7	2	Rapid override active	Rapid override	Direction keys +	-	Axis selector B	switch code A	* Deceleration X	Servo enable X
7	3	4th axis main axis		Data start	Reset		NC start	* Deceleration Y	Servo enable Y
8	4	C	Gear stage coding B	A	Spindle cw	Spindle enable	Feed enable	* Deceleration Z	Servo enable Z
8	5	* E-Stop	Read-in enable	Mirror image Y	X	Common axis inhibit	Operator panel lock	* Deceleration 4th axis	x Deceleration 4 th axos
8	6	Follow up mode spindle	Keyboard lock	Mirro image Z	Feed enable X	Set reference dimension X		Axis inhibit X	Follow up mode X
8	7	V-24 (RS232) inhibit	Keyboard switch over	Cycle lock	Feed enable Y	Set reference dimension Y		Axis inhibit X	Follow up mode Y
9	8	Clear distance to go	Cancel remaining no. of subrout. passes	Measuring	Feed enable Z	Set reference dimension Z		Axis inhibit Z	Follow up mode Z
9	9	Handwheel inhibit	Switch over to 3T	Resynchro. spindle	Feed enable 4th axis	Set reference dimension 4th		Axis inhibit 4th axis	Follow up mode 4th
9	10	Zero offset B	coding A						NC STOP
9	11							Screen darkening	Precontrol
OUTPUT INTERFACE 3M									
10	0	Act. direction cw	Spindle Speed limit	In command range	Stop		Traversing command Z	Y	X
10	1	Program active	NC alarm	NC ready 2	NC ready 1	Rapid	Threading	Measuring probe switched	Programmed stop M 00
10	2	Spindle in position	G 96 selected	V.24 (RS232) running	Switched to 3T	Reference point reached 4th axis	Reference point reached Z	Reference point reached Y	Reference point reached X
10	3	4 th positive	Travel direction commands Z positive	Y positive	X positive	Spindle speed at limit		G 38 selected	G 37 selected
11	4	M	Strobe signals S	T	H			M 02/M30 Reset	Rapid aux. function output
11	5	D (10**1)	C (10**1)	B (10**1)	BCD output A (10**1)	D (10**0)	C (10**0)	B (10**0)	A (10**0)
11	6	D (10**3)	C (10**3)	B (10**3)	BCD output A (10**3)	D (10**2)	C (10**2)	B (10**02)	A (10**2)

SINUMERIK SYSTEM 3		INPUT INTERFACE 3T				3T4C, 3TT4C			23.09.88
TESTING No.	Byte	Data bit number 7	6	5	4	3	2	1	0
7	0	Operating mode switch D	C	B	A	Feedrate/rapid override switch D	C	B	A
7	1	Key switch	Dry-run	Skip block	Single block	Block search	Spindle override switch C	B	A
7	2	Rapid override active	Rapid override	Direction keys X +	X -	Z +	Z -	* Deceleration X	Servo enable X
7	3		2nd software unit switch X active	Data start	Reset	Handwheel X	NC start	* Deceleration Z	Servo enable Z
8	4	Gear stage coding C	B	A	Spindle cw	Spindle enable	Feed enable		
8	5	* E-Stop	Data input enable	Mirror image Y	X	Common axis inhibit	Operator panel lock		
8	6	Follow up mode spindle	Keyboard lock		Feed enable X	Set reference dimension X		Axis inhibit X	Follow up mode X
8	7	V-24 (RS232) inhibit	Keyboard switch over	Cycle lock	Feed enable Z	Set reference dimension Z		Axis inhibit Z	Follow up mode Z
9	8	Clear distance to go	Cancel remaining no. of subrout. passes	Measuring					
9	9	Handwheel inhibit	Switch over to 3T	Resynchro. spindle					
9	10	Zero offset coding B	A	Tool path coding C	B	A	Select G39		NC STOP
9	11							Screen darkening	Precontrol
OUTPUT INTERFACE 3T									
10	0	Act. direction cw	Spindle Speed limit	In command range	Stop			Traversing command Z	X
10	1	Program active	NC alarm	NC ready 2	NC ready 1	Rapid	Threading	Measuring probe switched	Programmed stop M 00
10	2	Spindle in position	G 96 selected	V.24 (RS 232) running	Switched to 3T			Reference point reached Z	Reference point reached X
10	3			Travel direction commands Z positive	X positive	Spindle speed at limit	G 39 selected		
11	4	M	Strobe S	signals T	H			M 02/M30 Reset	Rapid aux. function output
11	5	D (10**1)	C (10**1)	B (10**1)	BCD output A (10**1)	D (10**0)	C (10**0)	B (10**0)	A (10**0)
11	6	D (10**3)	C (10**3)	B (10**3)	BCD output A (10**3)	D (10**2)	C (10**2)	B (10**2)	A (10**2)

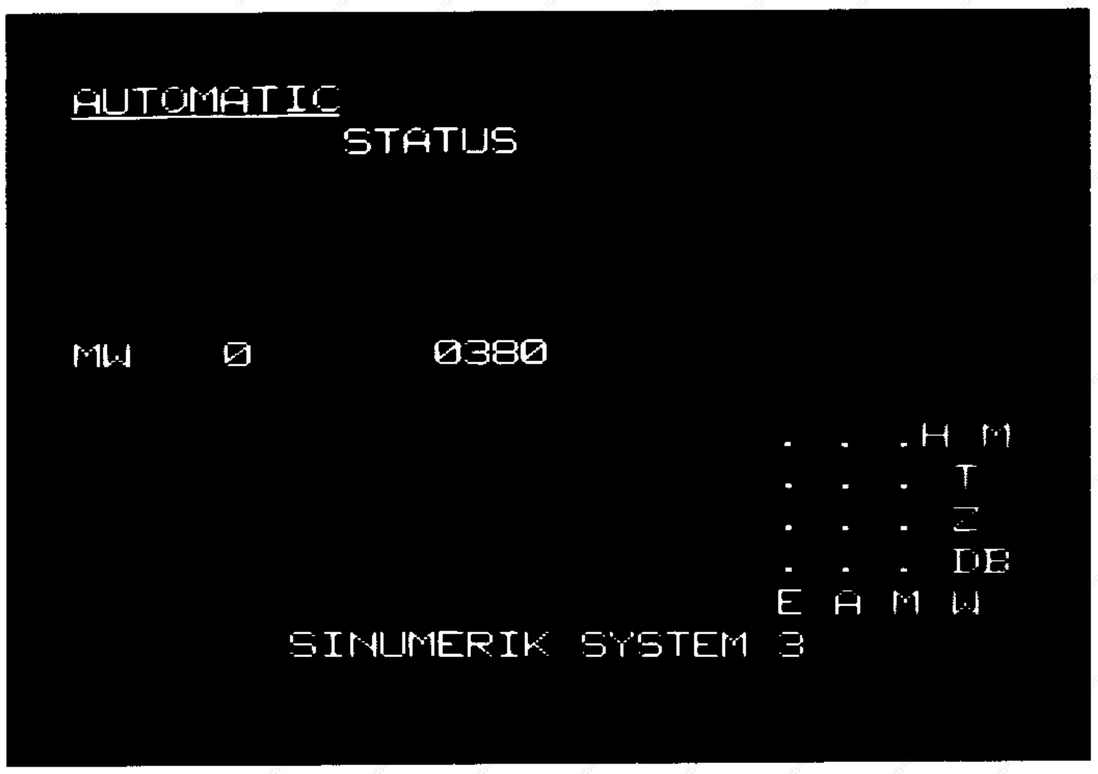
8.6.2.2 PLC status display

The control has test aids for service to display the statuses of all input and output signals between PLC and machine and the PLC-internal flags, timers, counters and data on the display of the control.

The input and output signals, flags and data can be entered via the operator panel of the NC.

Selecting the PLC Status Display

PC	Select the menu by switching to the customer display (No. 48)
O	The status display is selected by entering O.

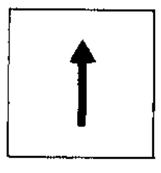


After selecting the PLC status display, the new designations of the NC address keys for status display are shown in the bottom right corner display.

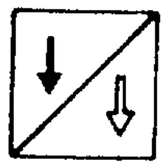
E = input signals (I)
A = output signals (Q)
M = flags (F)
T = timers
Z = counters (C)
W = word
DB = data block
H/M = changeover for hexa/bit pattern

General

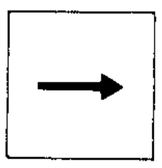
For the PLC status display keys 22 to 25 function as follows:



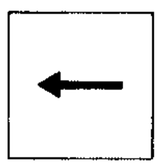
Decrement the byte or word address by 10



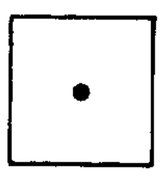
Increment the byte or word address by 10



Increment the byte or word address by 1



Decrement the byte or word address by 1



Increment the bit address by 1 Modulo 8

The decrementation/incrementation is executed automatically with a 1 Hz clock as long as the keys are pressed.

Illegal entries are underlined.
No transfer occurs.

Reading the Individual Input and Output Signals and Flags

E 0 · 4

Input I0.4 is selected.

Selection range 0.0 through 127.7

or

A 1 0 · 1

Output Q10.1 is selected.

Selection range 0.0 through 127.7

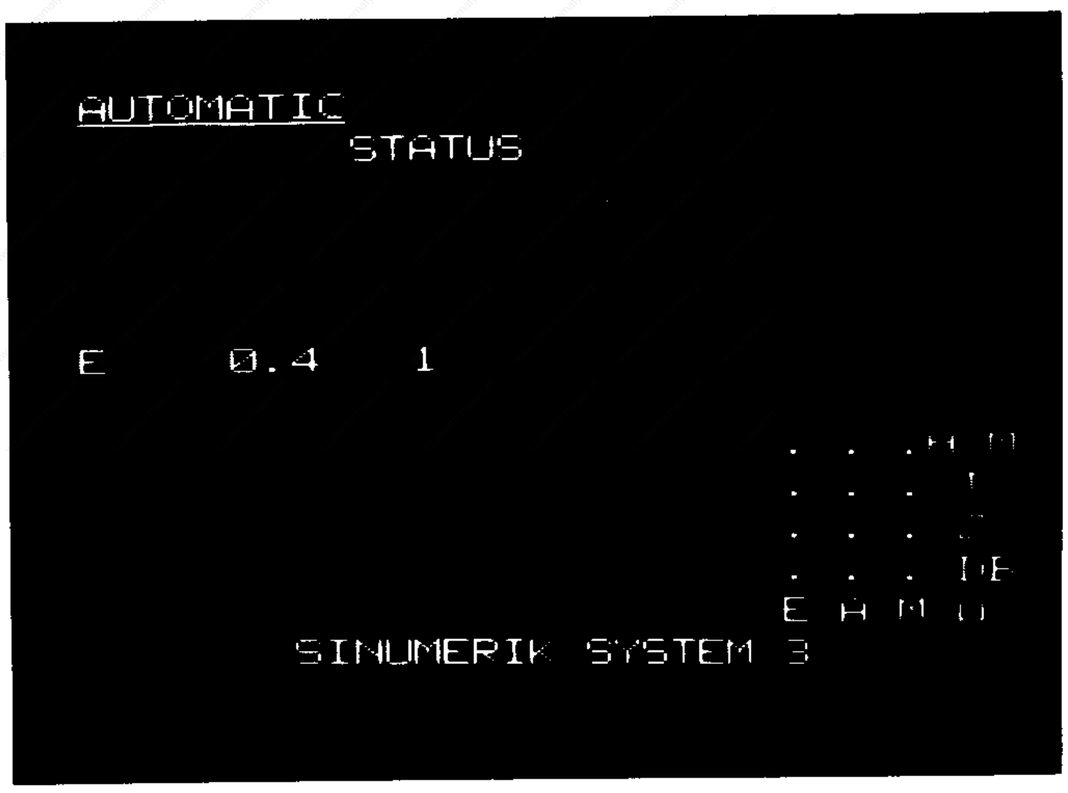
or

M 1 4 6 · 7

Flag F146.7 is selected.

Selection range 0.0 through 255.7

The input is cleared with key No. 29.



Reading Input and Output Signals and Flag Words

E W 6 3

Input signal IW63 is selected.
Selection range 0 through 126

or

A W 1 1 7

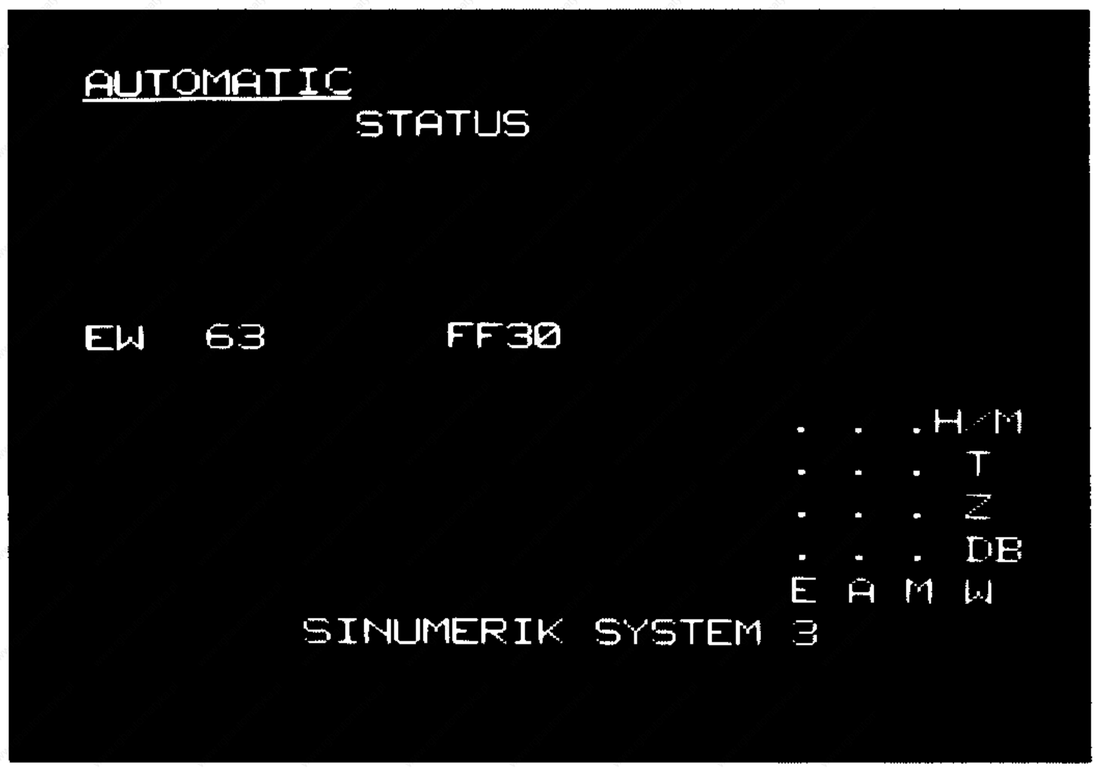
Output word QW117 is selected.
Selection range 0 through 126

or

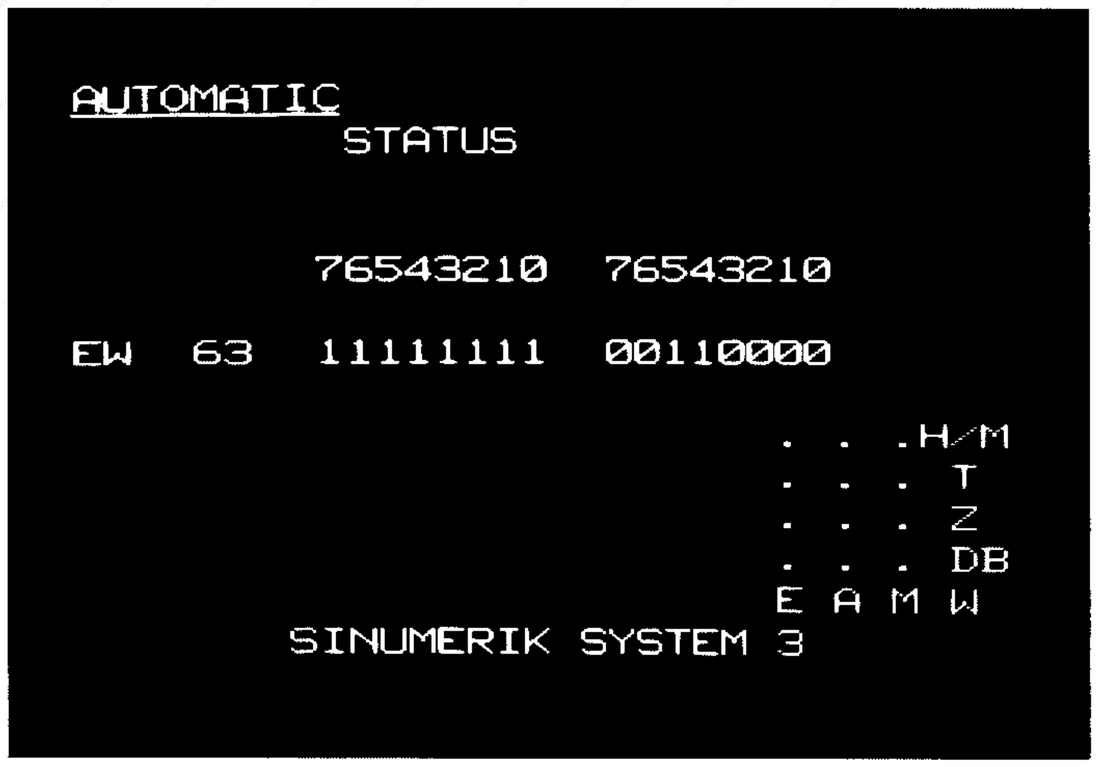
M W 3

Flag word FW3 is selected.
Selection range 0 through 254

The input is cleared with key No. 29.

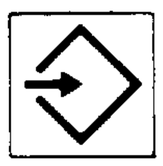


The word contents are displayed in hexadecimals after the H/M key has been pressed.



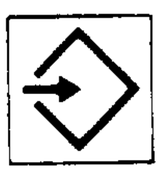
The word contents are displayed as bit pattern after the H/M key has been pressed.

Reading Timers and Counters

T 1 0 0 

Timer T100 is selected.

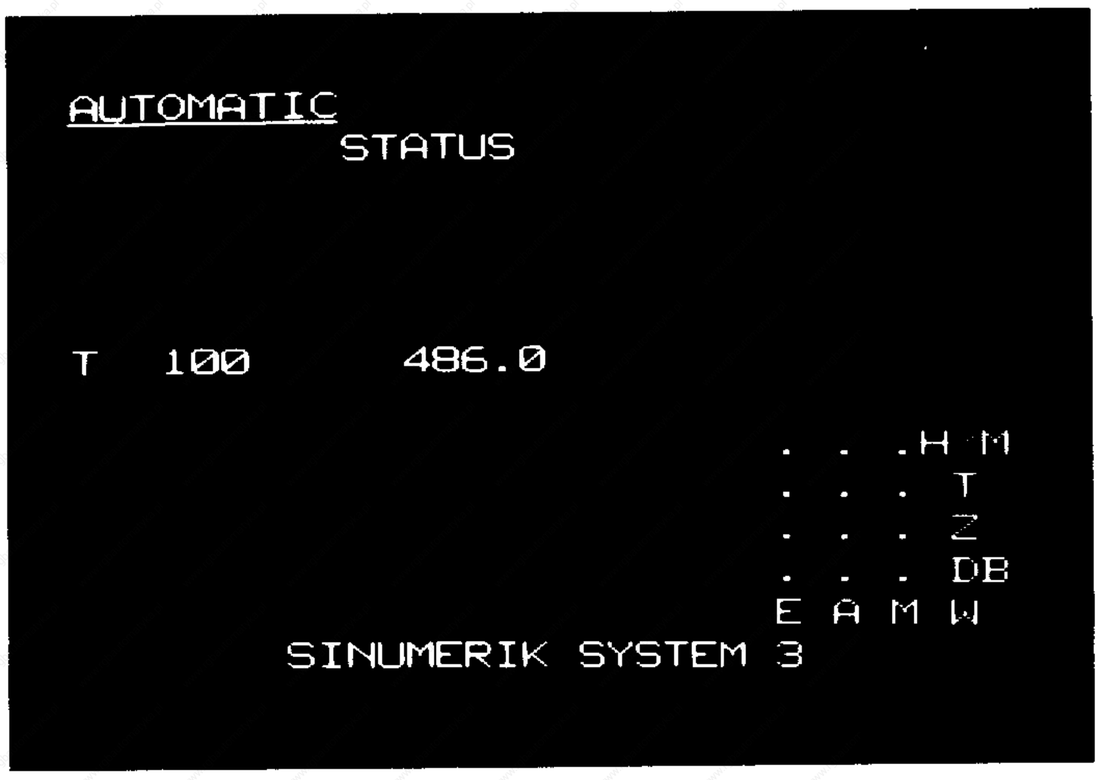
Selection range 0 through 127

Z 3 

Counter C3 is selected.

Selection range 0 through 127

The input is cleared with key Nr. 29.



The time is displayed in the STEP 5 programming language.

The counter value is displayed in BCD code.

Reading Data Words

DB 1 4 W 0

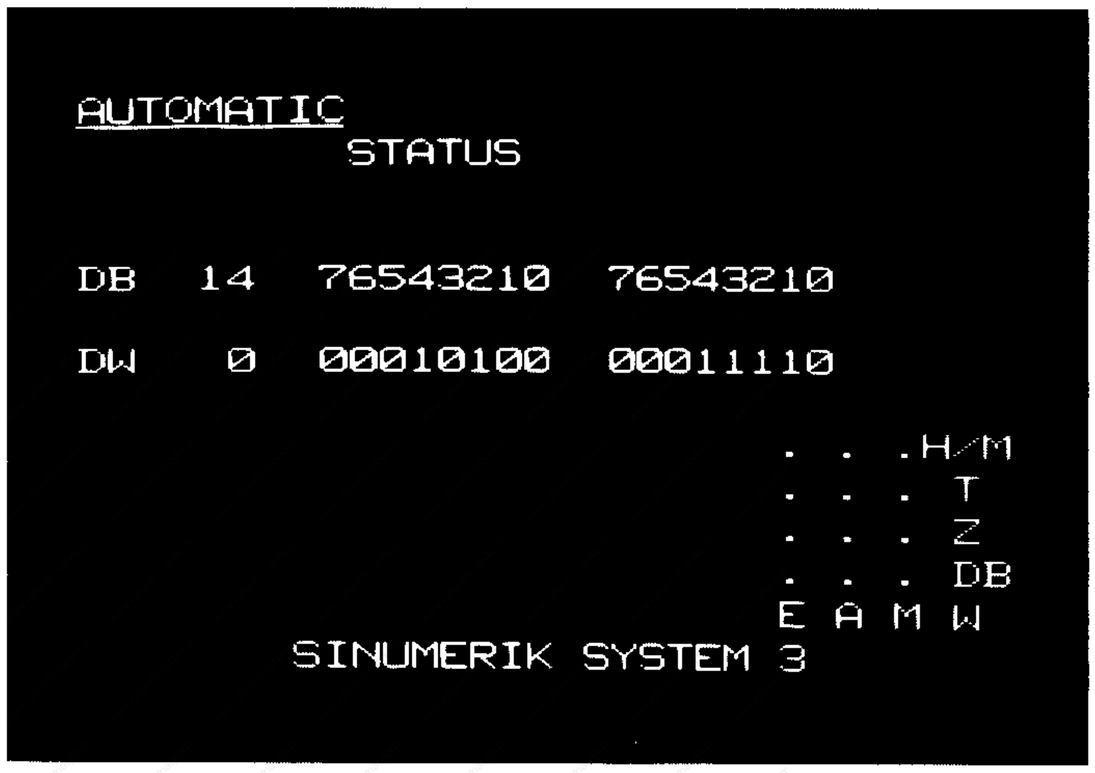
Data block DB 14 is selected.

Data word DW 0

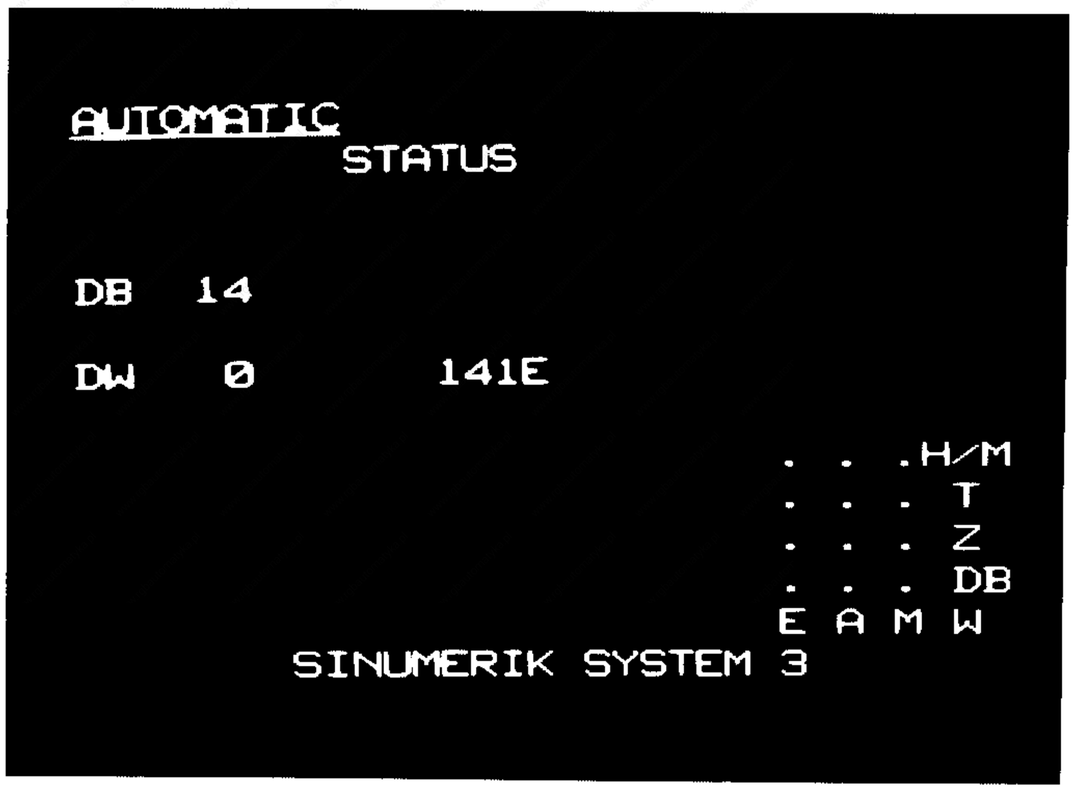
Selection range DB 1 through 255

DW 0 through 255

The input is cleared with key No. 29.



The word contents are displayed as a bit pattern after the H/M key has been pressed.



The word contents are displayed in hexadecimals after the H/M key has been pressed.

When data words are selected, a control-internal test is made to see whether the data word is defined or not.

If a data word is not defined, the display is:

Hexadecimal:	XX XX
Bit pattern:	XXXXXXXX XXXXXXXX.

Writing Individual Input and Output Signals and Flags

E 4 . 0 → Input I4.0 is selected.

Selection range 0.0 through 127.7

0 → or 1 → Input set to 0 or 1

or

A 1 0 0 . 0 → Output Q100.0 is selected.

0 → or 1 → Output set to 0 or 1

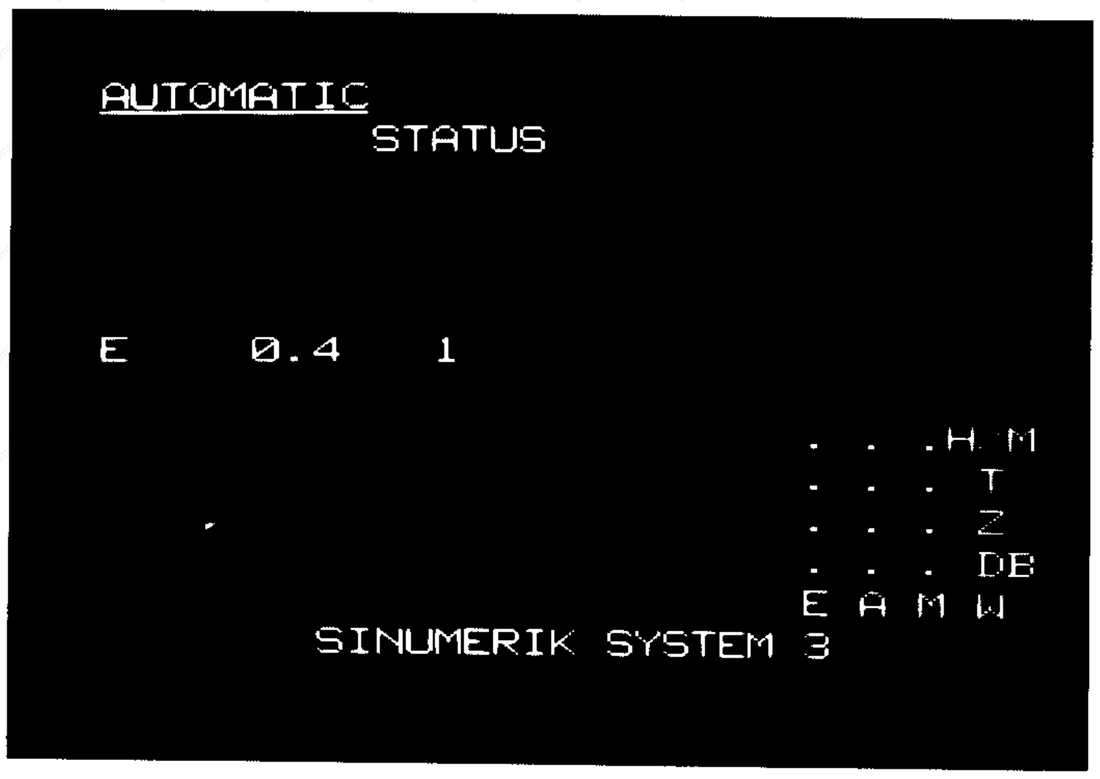
or

M 1 4 6 . 7 → Flag F146.7

Input range 0.0 through 229.7

0 → or 1 → Flag set to 0 or 1

Erroneous inputs can be cleared with key No. 29



When writing, the input/output signals and flags are set to 0 or 1 for one cycle. In the following cycle the input/output signal or flag can be changed again by means of the user program.

Writing Input/Output Signal and Flag Words

E W 1 2 3 

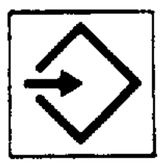
Input word IW123 is selected.

or

A W 9 7 

Output word QW97 is selected.

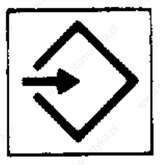
or

M W 8 

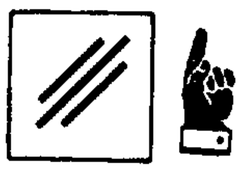
Flag word FW8 is selected.

Input range 0 through 228

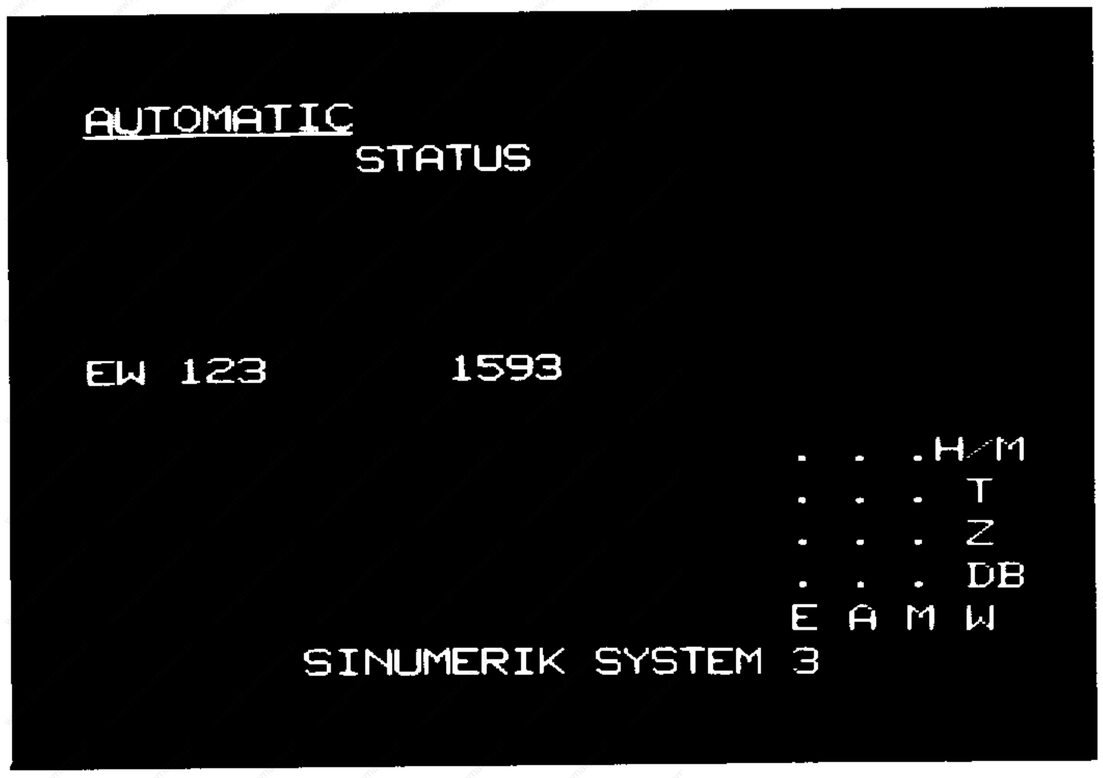
H/M

1 5 9 3 

Erroneous inputs can be cleared with key No. 29.



or



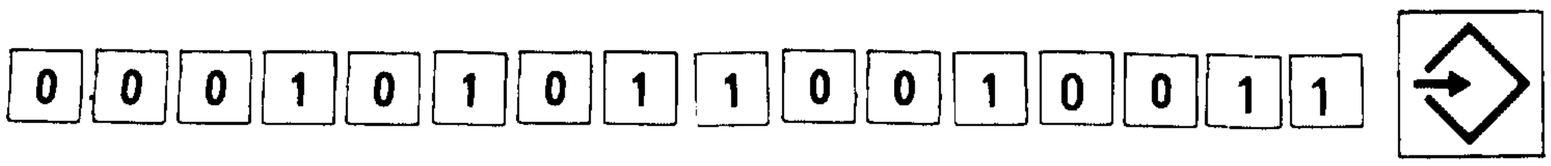
H/M

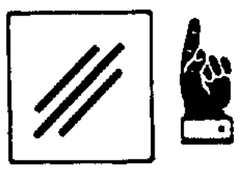
When entering in hexadecimals, key .0 through .5 should be pressed for A through F.

.0 = A .2 = C .4 = E

.1 = B .3 = D .5 = F

All inputs are entered right-justified.





Erroneous inputs can be cleared with key No. 29.

All inputs are entered right-justified.

Writing Data Words

DB 7 2 W 1 ⏎

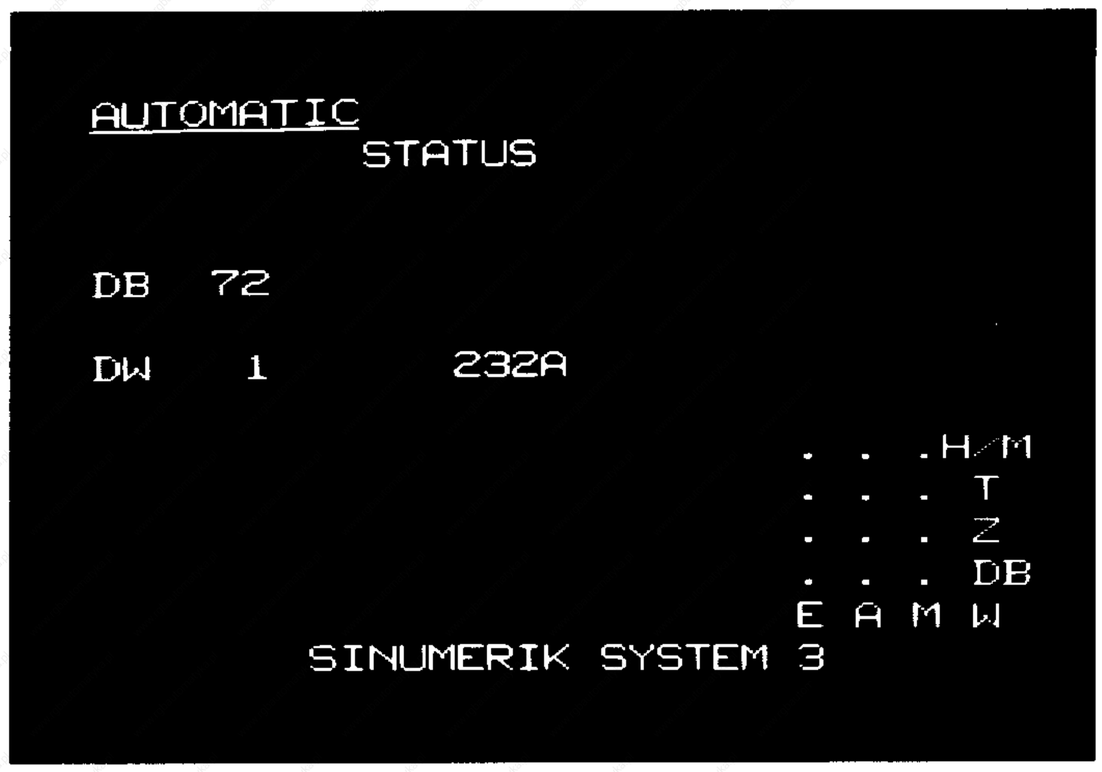
H/M

Switch to hexadecimal display.

2 3 2 . 0 ⏎

Erroneous inputs can be cleared with key No. 29.

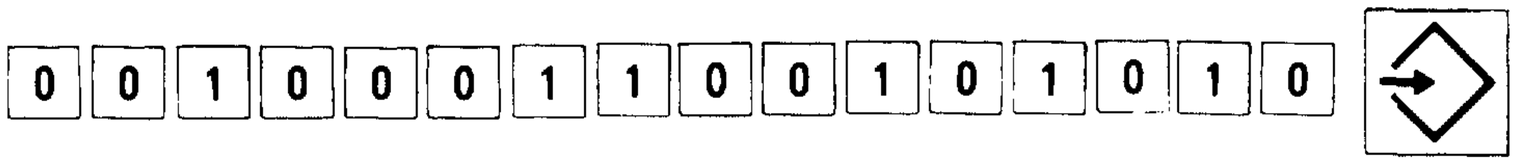
or

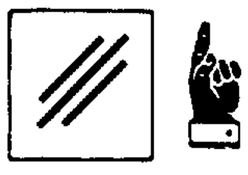


All inputs are entered right-justified.
When entering in hexadecimal keys .0 through .5 should be pressed for A through F.

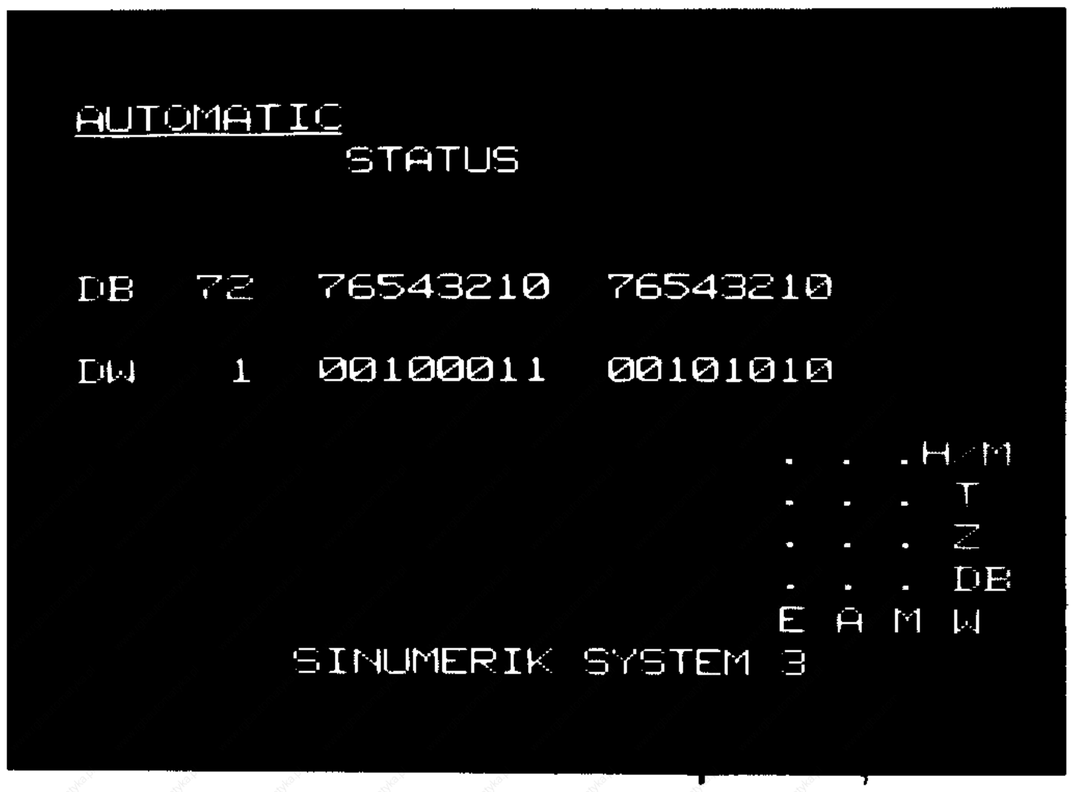
.0 = A	.2 = C	.4 = E
.1 = B	.3 = D	.5 = F

Switch to bit pattern display.



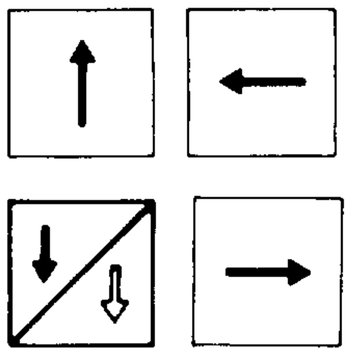


Erroneous inputs can be cleared with key No. 29.



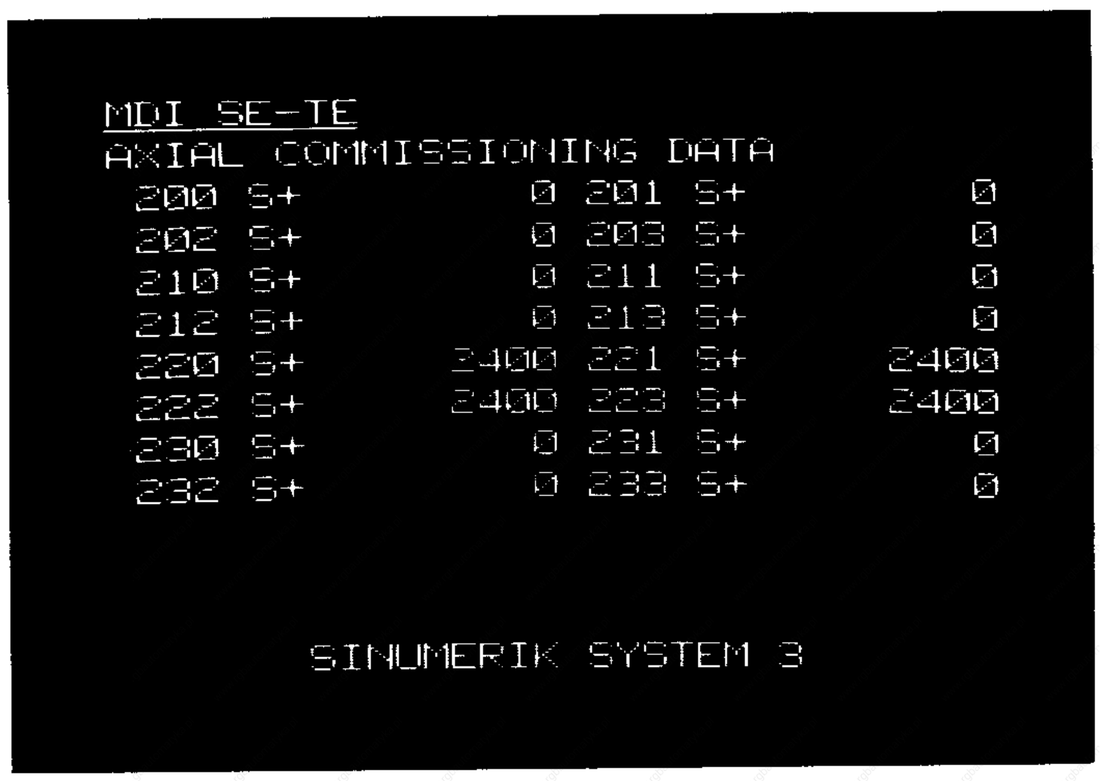
All inputs are entered right-justified.

8.6.3 NC Machine Data



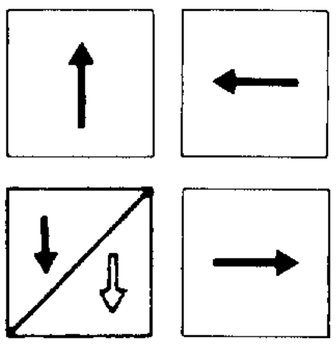
Select display numbers 100 through 755 with the "Page" and "Cursor" keys (No. 22, 23, 24, 25)

Machine data are determined and entered during the start-up of the machine.



Axis-specific machine data	(100-279)
General machine data	(280-399)
Machine data bits	(400-755)

8.6.4 Service Values



Select display numbers 800 through 861 with the "Page" and "Cursor" keys (No. 22, 23, 24, 25).

No input possible.

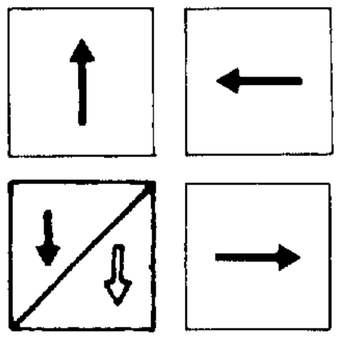
```
MDI SE-TE
COMMISSIONING DISPLAY DATA
 800 S+          22 801 S-          33
 802 S+           0 803 S+           0
 810 S+           0 811 S+           0
 812 S+           0 813 S+           0
 820 S+           4 821 S-           3
 822 S+           0 823 S+           0
 830 S+           0 831 S+           0
 832 S+           0 833 S+           0
 840 S+           0 841 S+           0
 842 S+           0 843 S+           0

        SINUMERIK SYSTEM 3
```

The following values are displayed:

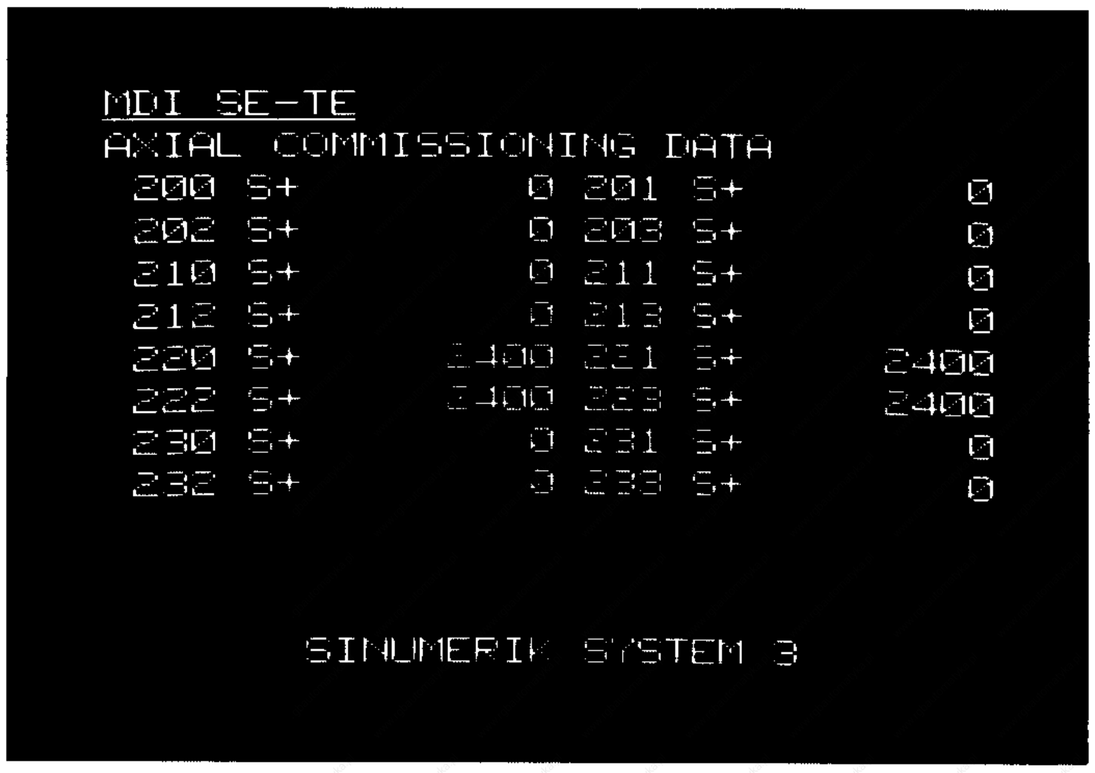
Ident. No.	Axis		Display
800 S	X	X	Following error
801 S	Z	Y	Following error
802 S	-	Z	Following error
803 S	-	4th.	Following error
810 S	X	X	Actual value (measuring system)
811 S	Z	Y	Actual value (measuring system)
812 S	-	Z	Actual value (measuring system)
813 S	-	4th.	Actual value (measuring system)
820 S	X	X	Set speed
821 S	Z	Y	Set speed
822 S	-	Z	Set speed
823 S	-	4th.	Set speed
830 S	X	X	Part actual value
831 S	Z	Y	Part actual value
832 S	-	Z	Part actual value
833 S	-	4th.	Part actual value
840 S	X	X	Contour deviation
841 S	Z	Y	Contour deviation
842 S	-	Z	Contour deviation
843 S	-	4th.	Contour deviation
850 S	X	X	Measured KV factor
851 S	Z	Y	Measured KV factor
852 S	-	Z	Measured KV factor
853 S	-	4th.	Measured KV factor
860 S	-	-	Set speed (Spindle)
861 S	-	-	Spindle position

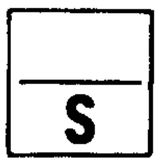
8.6.5 Setting the Drift Compensation

Voltage drift will occur in the analog components of the position control loop as a result of temperature changes. This drift can lead to deviations from the set values, and can be counteracted by setting a drift compensation.



Select display numbers 200 through 233 by using the "Page" and the "Cursor" keys (No. 22, 23, 24, 25)





The drift value is transferred automatically to the display. This setting of the drift compensation is not protected by the machine data lock switch. The drift compensation applies to the axis selected with the cursor.

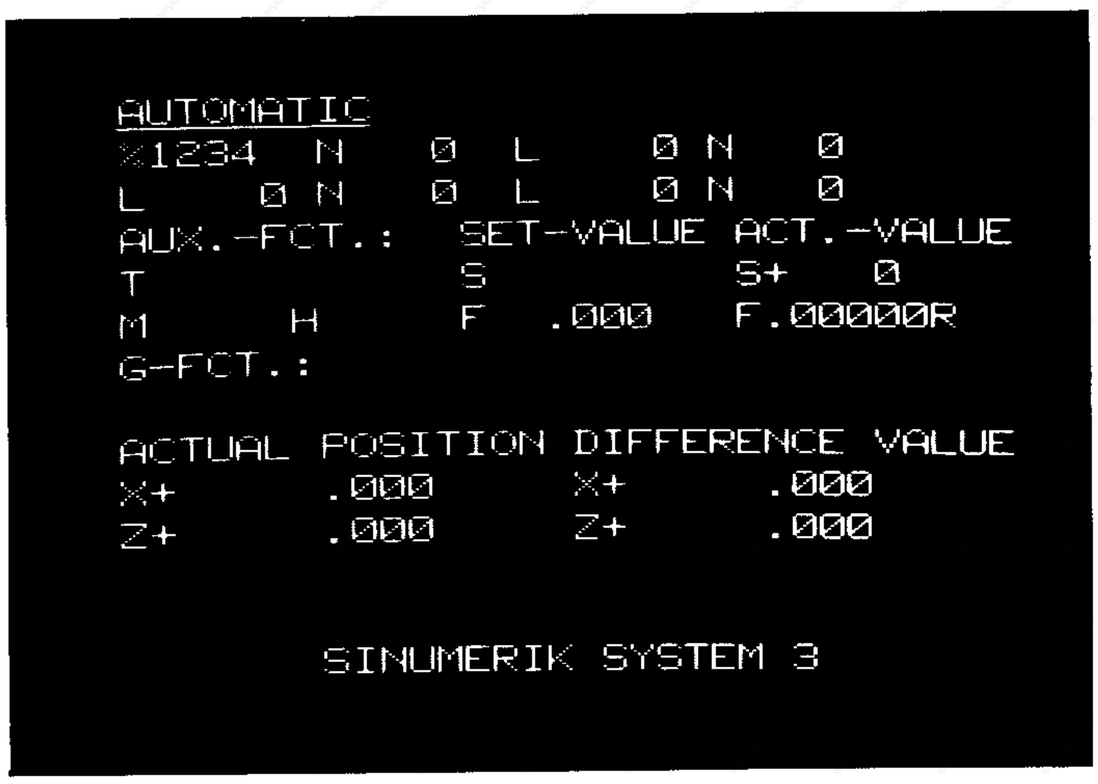
Page Number	Axis	
Cursor Position	3T	3M
230	X	X
231	Z	Y
232	-	Z
233	-	4th.

9. Automatic Mode (AUT)

The control calls up the program blocks one after the other and executes them. After a block has been executed, the next prepared block is read and executed.

"Prepared block" means that the axis movements programmed in this block have been calculated to include "settable zero offsets" and "tool offsets". The only source of data for this process is the NC internal program memory.

Automatic mode (selector switch No. 31)

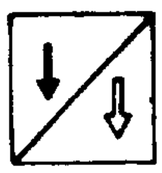


Operating mode (Automatic)

Display of the actual program with all nestings.

Display of the subroutines with program numbers and processing pointers (see Section 1.3).

For dual display (page 1 to 6), the basic display is split into two pages.



Selection via page key

```
AUX.-FCT.:  SET-VALUE ACT.-VALUE
T     D 0   S         S+    0
M           F     0   F   .000M
```

T, D, M, H Functions (AUXILIARY FUNCTIONS:)

T for the 3T: Tool number with offset number (2+2 decades)

T for the 3M: Tool number (4 decades)

D offset number D is only shown here for the 3 M

M programmed additional function

H programmed auxiliary function

S, F (SET VALUES)

S programmed spindle speed

F programmed feed rate
(dimensions as for the actual feed rate)

S, F (ACTUAL VALUES)

S current spindle speed

F current feedrate
(dimension "R" = per revolution)
(dimension "M" = per minute)

```
G-FKT.: G01  .   G04 G40     G54
        G71 G90 G95
```

G (G FUNCTIONS:)

Display of all current G functions

ISTWERTE		SOLL-IST-DIFF.	
X+	.000	X+	.000
Z+	.000	Z+	.000

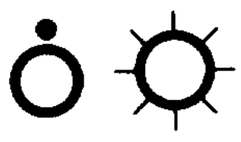
Actual Values (ACTUAL POSITION)

Display of actual values for all axes (machine- or workpiece-oriented actual value system can be selected via "Bit-oriented user data", see Section 8.4.).

Distance to go (DIFFERENCE VALUE)

The distance still to be traversed to the end point of the block is shown for all axes.

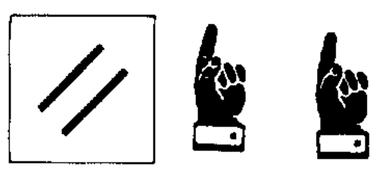
The feedrate programmed has the same dimension as the actual feedrate.



If the Feed Hold is on (No. 2),
the programmed spindle speed S, and
the tool number T, and
the programmed auxiliary functions M and H,
can be transferred (overstored)

For processing, the program pointer must be set to the desired value.

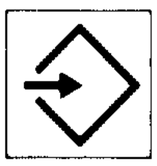
Operation Sequence



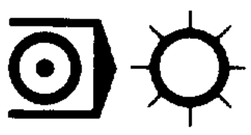
only when control is not in the "Reset" state

Automatic mode (selector switch No. 31)

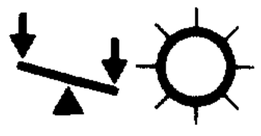
% 20 Enter the desired program number



The program is started with the "Cycle Start" key (No. 44)



"Program active" lamp lights up (No. 1)



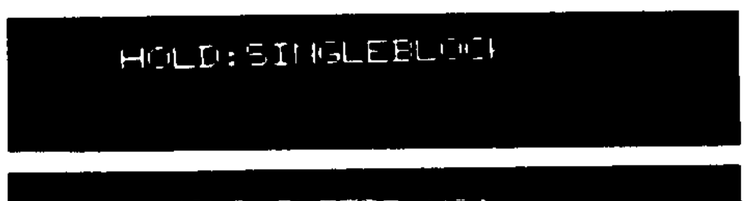
During operation, as long as the machine axes are in motion, the "Position not yet reached" lamp (No. 3) stays lit.

9.1 Influencing automatic mode

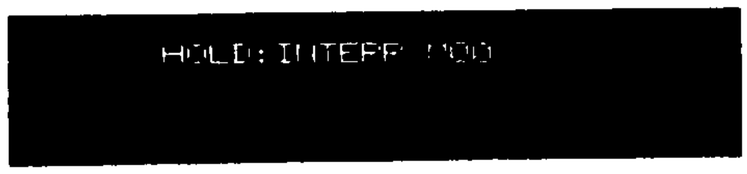
Displays

The submodes are displayed in line 16.

Automatic mode interrupted

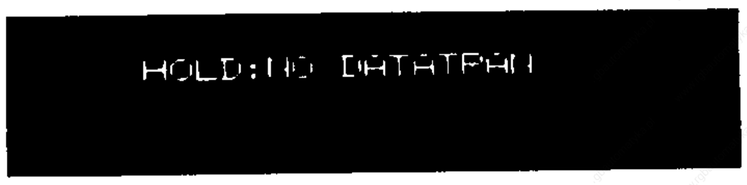


Single block executed

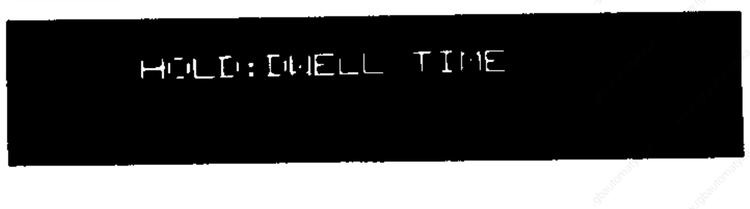


Programmed program interruption MOO

The next block is read, prepared, and executed by pressing the "Cycle Start" key (No. 44).



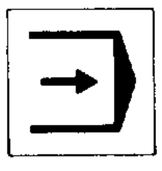
Data input not enabled



Dwell time active

Operation Submodes

Single (No. 40)

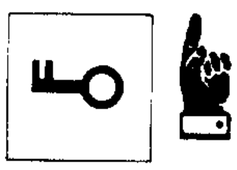


Each time the "Cycle" key is pressed, a single block is called up, prepared, and executed.

Skip (No. 41)

Blocks marked "/" are not used in the program sequence (Skippable block)

9.2 Tool Offset, Zero Offset, Machine Data Displays



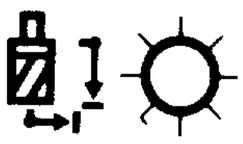
Enable keyswitch (No. 38) for input

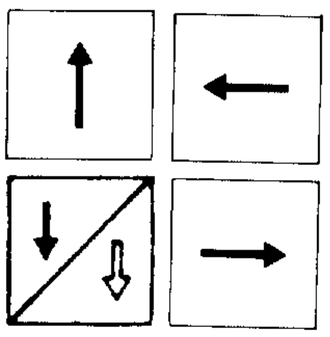
depending on machine datum

For the 3T:

Select tool offset (No. 6)

For the 3M:





Select tool number

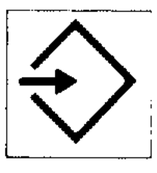
e.g.
B, 999

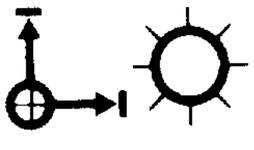
A tool wear value can also be entered while the program is running (can be interlocked using the keyswitch).
Input of wear value, max. ± 0.999 mm
± 0.0999 inch

and / or

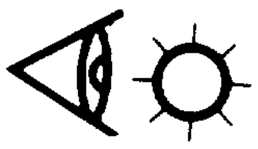
e.g.
B 10 000

Tool offset values can also be entered while the program is running (can be interlocked using the keyswitch).





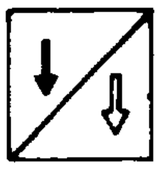
Zero offset display (No. 7)
Input not possible.



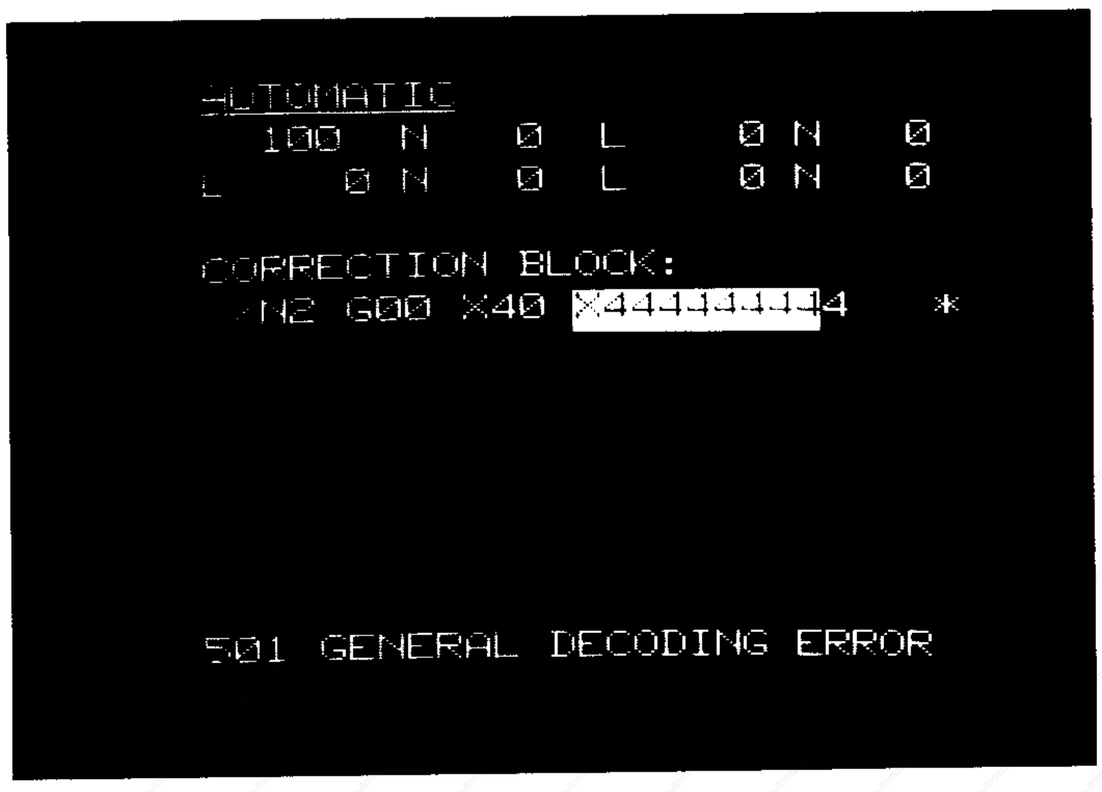
Test display (No. 8) (Machine data)
Input not possible.

9.3 Correction Block Display

This display shows the current program block, which can still be modified.



Selection via page key



Display of the actual program with all nestings.

Display of subroutines with program numbers and processing pointer.

Input is not possible.

Correction Block

The incorrect characters detected during decoding are displayed in inverse video.

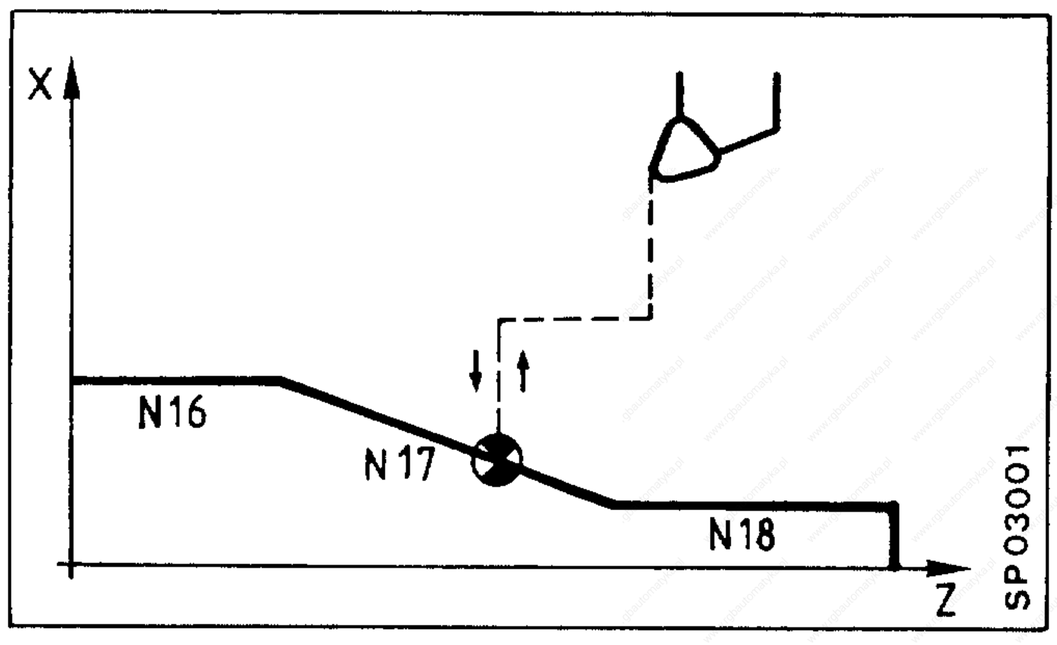
The incorrect words can be corrected in the MDI Part Program mode (see Section 7).

9.4 Resuming Automatic Mode after an Interruption

9.4.1 Overview

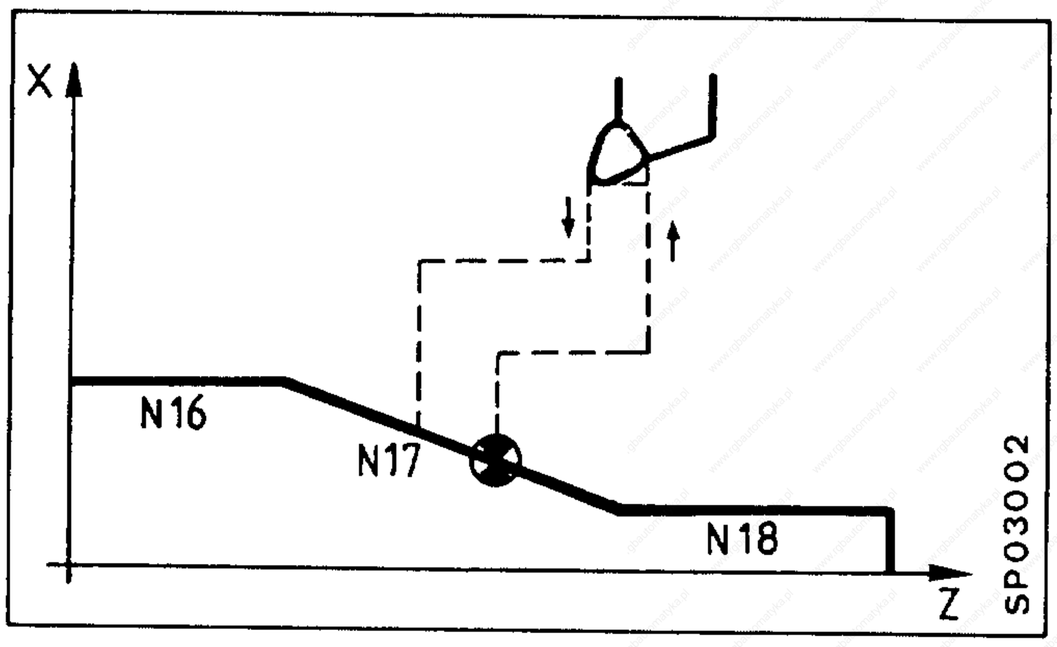
There are four ways of returning to automatic mode after a program has been interrupted.

a) Retracting for Tool Inspection



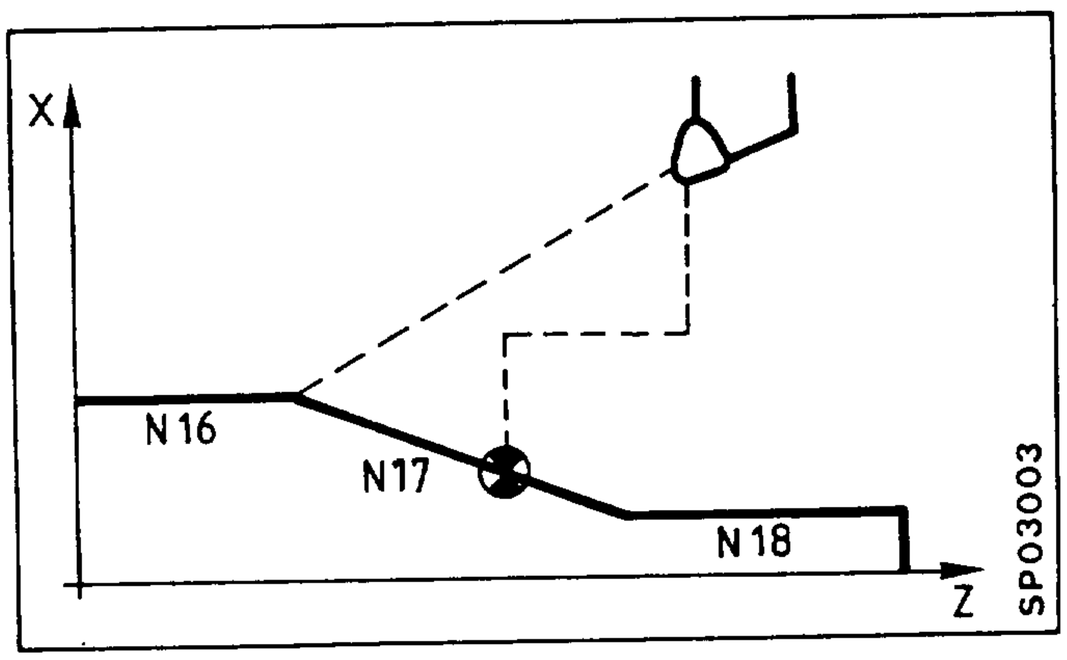
The tool is brought incrementally back to the position of interruption. Tool offsets remain unchanged.

b) Scratching



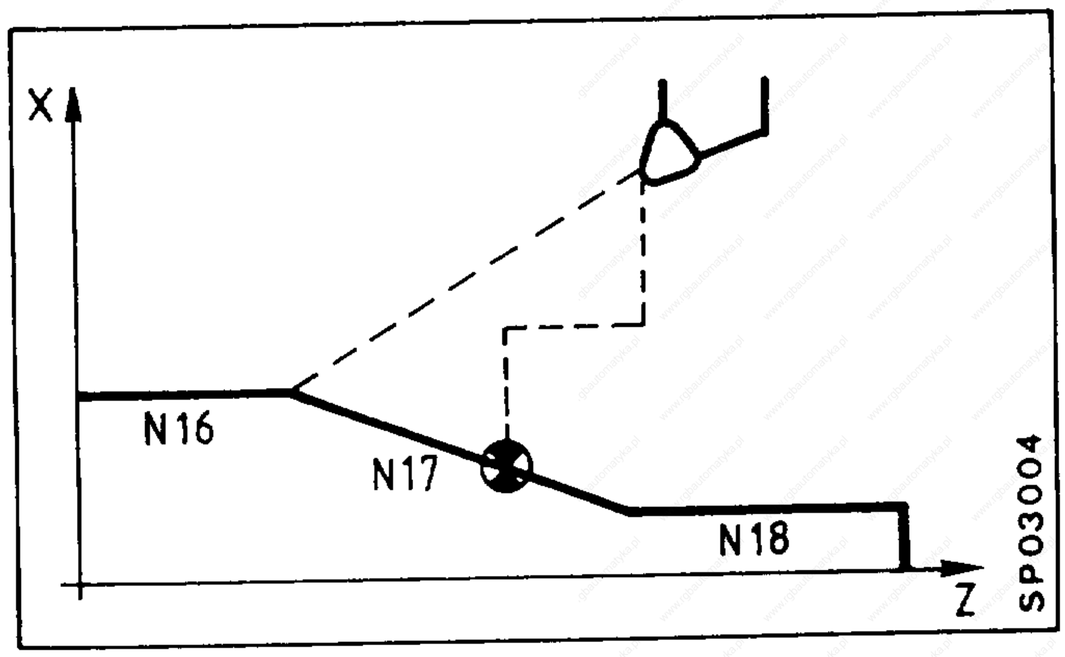
Scratching within the interrupted block in jog mode.
Tool length may vary. Cutter radius compensation remains unchanged.

c) Block Search to Interrupted Block



The control positions the tool with linear interpolation automatically at the start of the interrupted block. (SINUMERIK 3T)
Tool offsets may vary.

d) Block Search to Arbitrarily Selected Block



The control positions the tool with linear interpolation automatically at the end point of the preselected block.
(In this example, the preselected block is N16.)
Tool offsets may vary.

9.5 Program Interruption through Set-up

9.5.1 Retracting for Tool Inspection

The Automatic (AUT) operating mode can be interrupted by means of Set-up. (JOG; INT)

! Hold Auto Interr !	Interruption of Automatic Mode

The program pointer and the status pointer are not changed. The tool is moved incrementally away from the contour for inspection, and then is brought back to the point of interruption.
If the machine is moved in JOG mode, the operator must remember the interrupt position values.
After returning to automatic mode, machining is continued with "Cycle Start".
Other interruptions lead to Alarm 511.

If the axes are traversed with JOG or INC and a return is not made to the point of interruption, the absolute block end of the programmed axes is approached after resuming with "Cycle Start". Axes that have not been programmed can be corrected only after they have been programmed.

This also applies to the end of single blocks.

The tool nose radius compensation/cutter radius path compensation cannot be changed.

9.5.2 Scratching

The Automatic (AUT) mode can be interrupted by means of Set-up (JOG; INC).

Hold: Auto Interr	Interruption of Automatic mode

The tool is moved away incrementally or with JOG from the contour for replacement, e.g. when the tool breaks. After the new tool length compensations have been entered (tool nose radius/cutter radius remain unchanged), the new tool is brought to an arbitrary point on the contour within the interrupted block.

After returning to Automatic mode, machining is continued with "Cycle Start".
Other interruptions lead to Alarm 511.

If the axes are not returned to the point of interruption, the absolute block end of the programmed axes is approached after resuming with "Cycle start". Axes which have not been programmed are corrected only after they have been programmed.

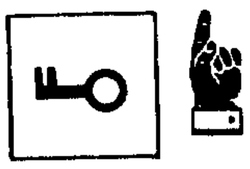
This also applies to the end of single blocks.

9.6 Block Search in Automatic Mode

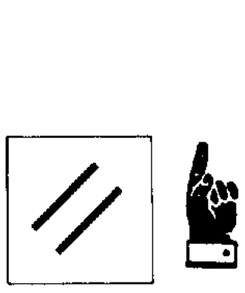
9.6.1 Block Search for Interrupted Block

After an interruption of Automatic mode, e.g. because of tool breakage, the tool can be positioned automatically to the beginning of the interrupted block by means of the "Block search for interrupted block" function.
This function is particularly helpful in cases where the contours of the interrupted block are not clearly visible to the operator.
In such cases, the SINUMERIK 3T automatically positions the tool with linear interpolation at the starting of the interrupted block.
During block search, all new tool offsets, including tool nose-and cutter radius compensations, are taken into account by the control.

Operation Sequence with Operator Guiding

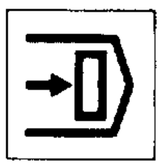


Key switch (No. 38) must be unlocked if the input is interlocked with the key switch.
(Machine datum)

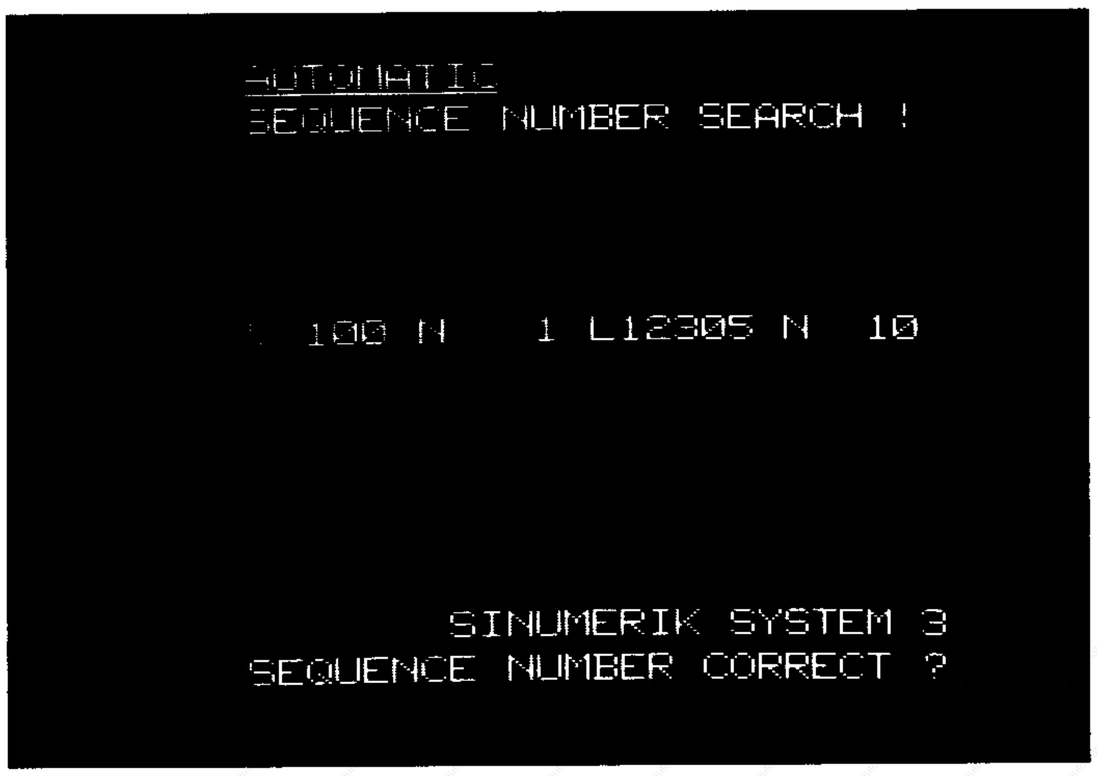


"Reset" key (No. 12), if the controller is not in Reset state.

Select Automatic mode (selector switch No. 31)



Block search key (No. 39)

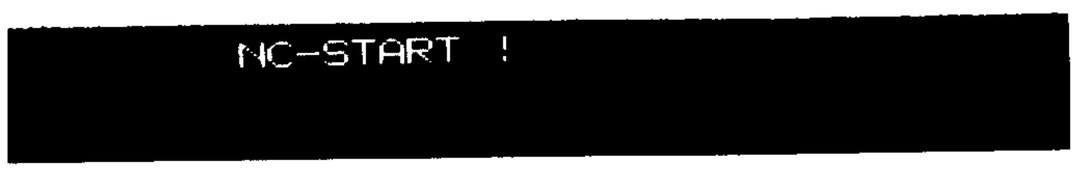


The last active program state is displayed here; the processing pointer (block number) shows the last completely processed block, including the first nesting level.
The text for the operator prompting is displayed in line 15.

YES

Block search for the displayed program state.
The program is continued from the beginning of the block which was interrupted.

The above display appears during the search.
The control calculates the contour up to the displayed program block, without axis movement.



The above display appears after block search; it prompts the pressing of the "Cycle Start" key (No. 44).

"Cycle Start" initiates automatic traversing to the beginning of the interrupted block (displayed program state), and then the program is continued. Skippable blocks are skipped in block search, if the "Skip" switch (No. 41) is on.
The Automatic basic display appears again.

Depending on the machine datum, the miscellaneous and auxiliary functions (M, S, T, H) are locked, output, or collectively output with the "Cycle Start" key (No. 44) (last M, S, T, H word).
Missing miscellaneous and auxiliary functions can be transferred (overstored) after "Cycle Start", during feed hold.

9.6.2 Block search for Arbitrarily Selected Block

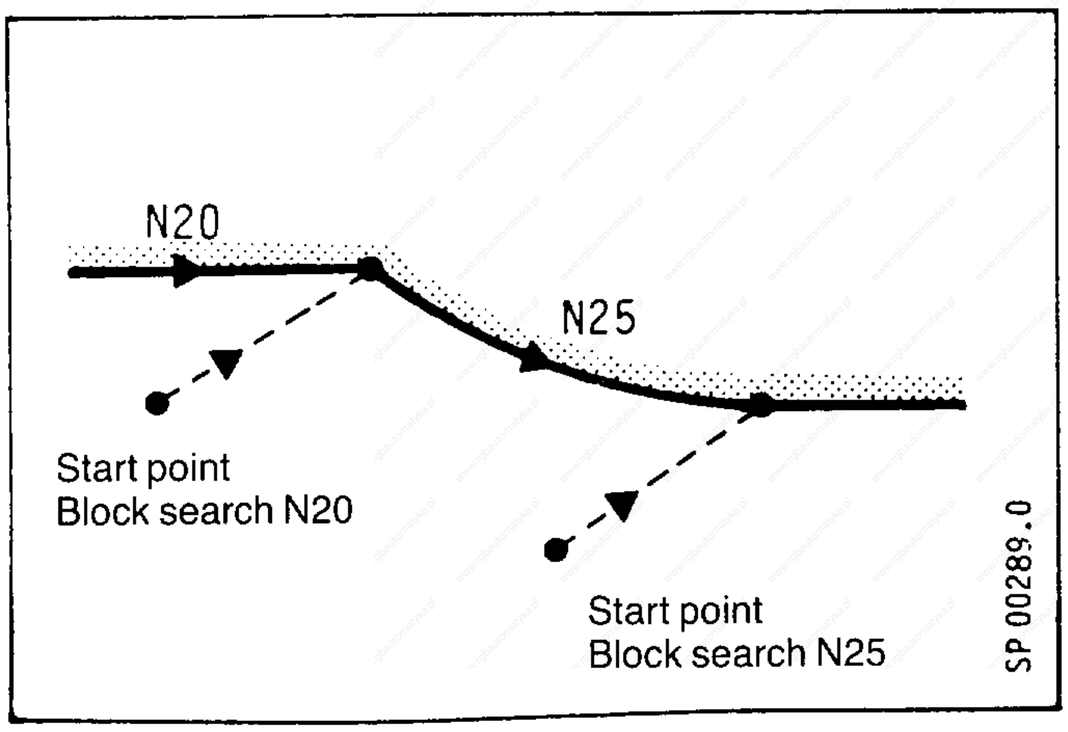
This block search function allows a program to start an arbitrary point in a part program or subroutine in the fist nesting level.
During the block search all the calculations are executed (tool dimensions, zero offsets, parameter calculations, intersection-, cutter radius-, and tool nose compensations).

The control automatically positions the tool at the end of the preselected block.

Depending on the type of part program generation, two operation sequences are possible for block search:

1. The first position information for all axes is programmed with G90 (the control thus knows the workpiece zero)

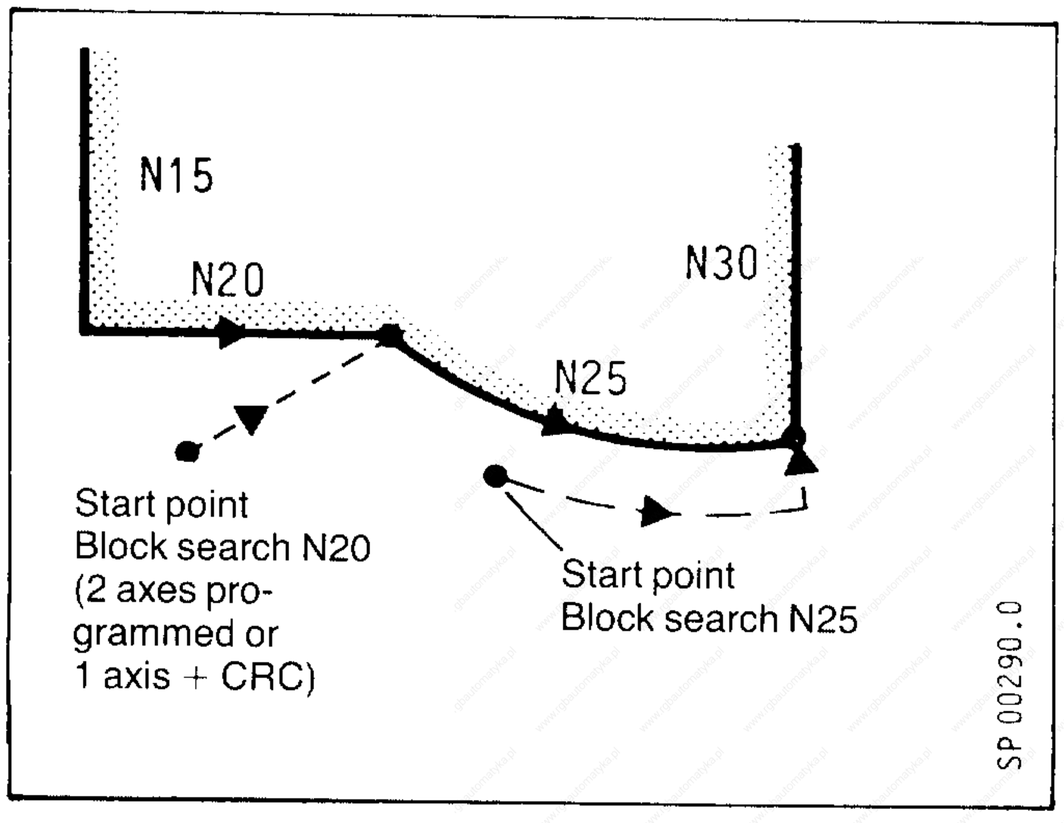
3T: In a Setting mode bring the tool close to the contour in the direction of machining, before the end of the preselected block.



3M: If the preselected block contains:

- 1 items of position data and if cutter radius compensation is active, or
- 2 items of position data

then in Setting-up mode, the tool should be brought close to the contour in the direction of maching, before the end of the preselected block.



If the preselected block contains:

- 1 item of position data,

the tool should be positioned, in a Setting-up mode, on the contour in the direction of machining, before the end of the preselected block.

If the preselected block contains:

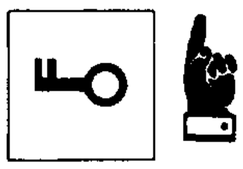
- no position, or
- 1 item of position data and no cutter radius compensation,

the axes are positioned as soon as they are programmed in subsequent blocks.

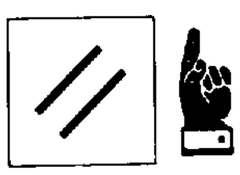
2. The position information for all axes is programmed with G91 (the control does not know the position of the workpiece zero)

Before block search, all the programmed axes must be in positions corresponding to the start of the program.
After block search is completed, continue as described in 1.

Operation Sequence with Operator Dialogue

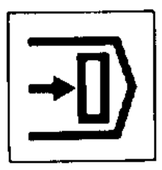


Key Switch (No. 38) must be unlocked if the input is interlocked with the key switch (machine data)

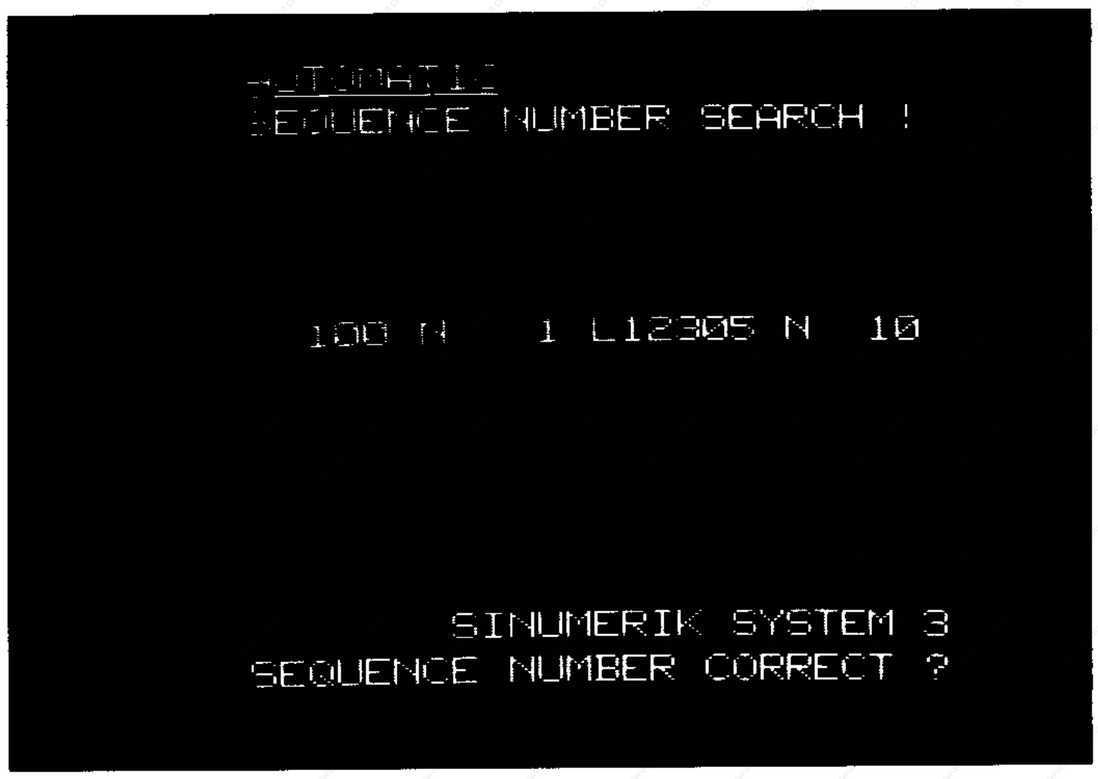


"Reset" key (No. 12), if the controller is not in Reset state.

Select Automatic operating mode (selector switch No. 31)



"Block search" key (No. 39)



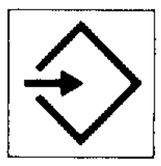
The last active program state is displayed here; the processing pointer (block number) shows the last completely processed block, including the first nesting level.
The text for the operator prompting is displayed in line 15.

Selection of a block other than that displayed.

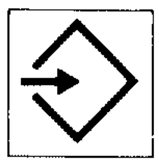
Line 15:

Operator prompting for input of the block search destination (e.g. N5, L1101, N10, selected main program 100).

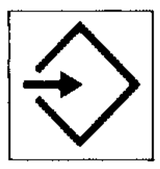
N5 Entry of block number



L1101 Entry of subroutine number including number of passes.



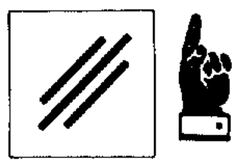
N10 Entry of block number in the subroutine



Block search to the displayed program state.

For continuation of the operation, see:

"Block search for interrupted block" (Section 9.6.1)



Termination of block search destination input

(no block search into the subroutine).

9.6.3 Block search without calculation of the paths (option B52)

The function permits

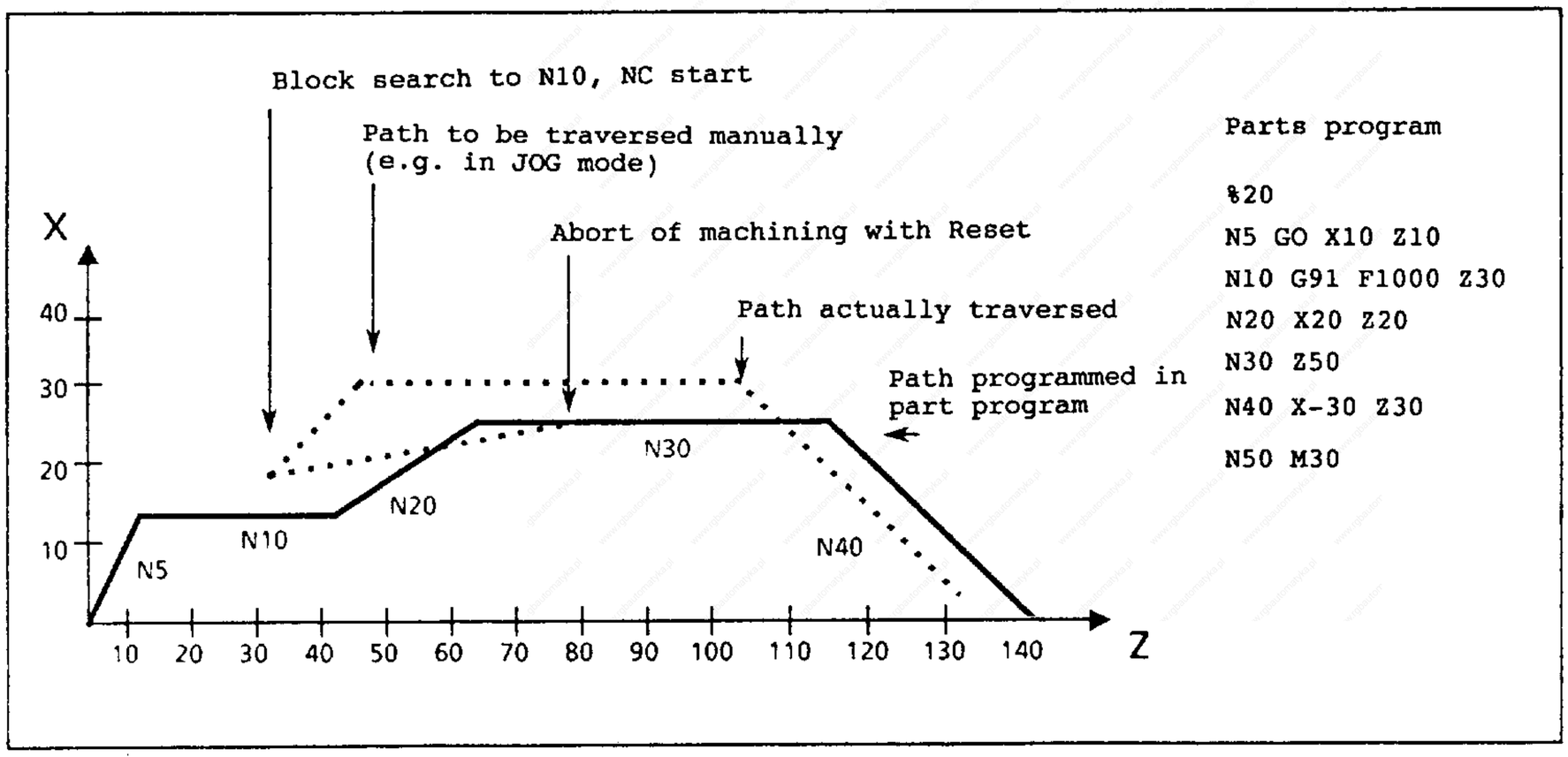
- the continuation of an interrupted subroutine from any actual machine position.

- the continuation of part program processing from the actual machine position after a block search as if the actual machine position has been reached with the end position of the block to which the block search had gone.

Application:

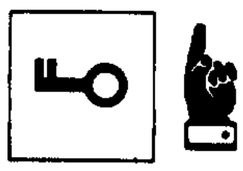
In automatic mode and a running part program, the axes can be traversed (e.g. by switching to JOG mode and pressing the direction keys). The part program can then be continued in automatic mode, if the following NC blocks only contain incremental dimensions.

Example:



9.7 Program Test

9.7.1 Dry Run



Unlock keyswitch (No. 38) if necessary, depending on machine data

Dry (No. 42)

Dry run feedrate
if active in automatic mode has the following effect:

- The programmed feedrates are suppressed by the control.
  Dry run feedrate is active, in mm/min (TE 375)

- Rapid traverse remains unchanged.
  Feed and rapid traverse override switches are active.
- For threads and feedrates per revolution, the control also switches over to the stored dry run feedrate.

9.7.2 Syntax Test

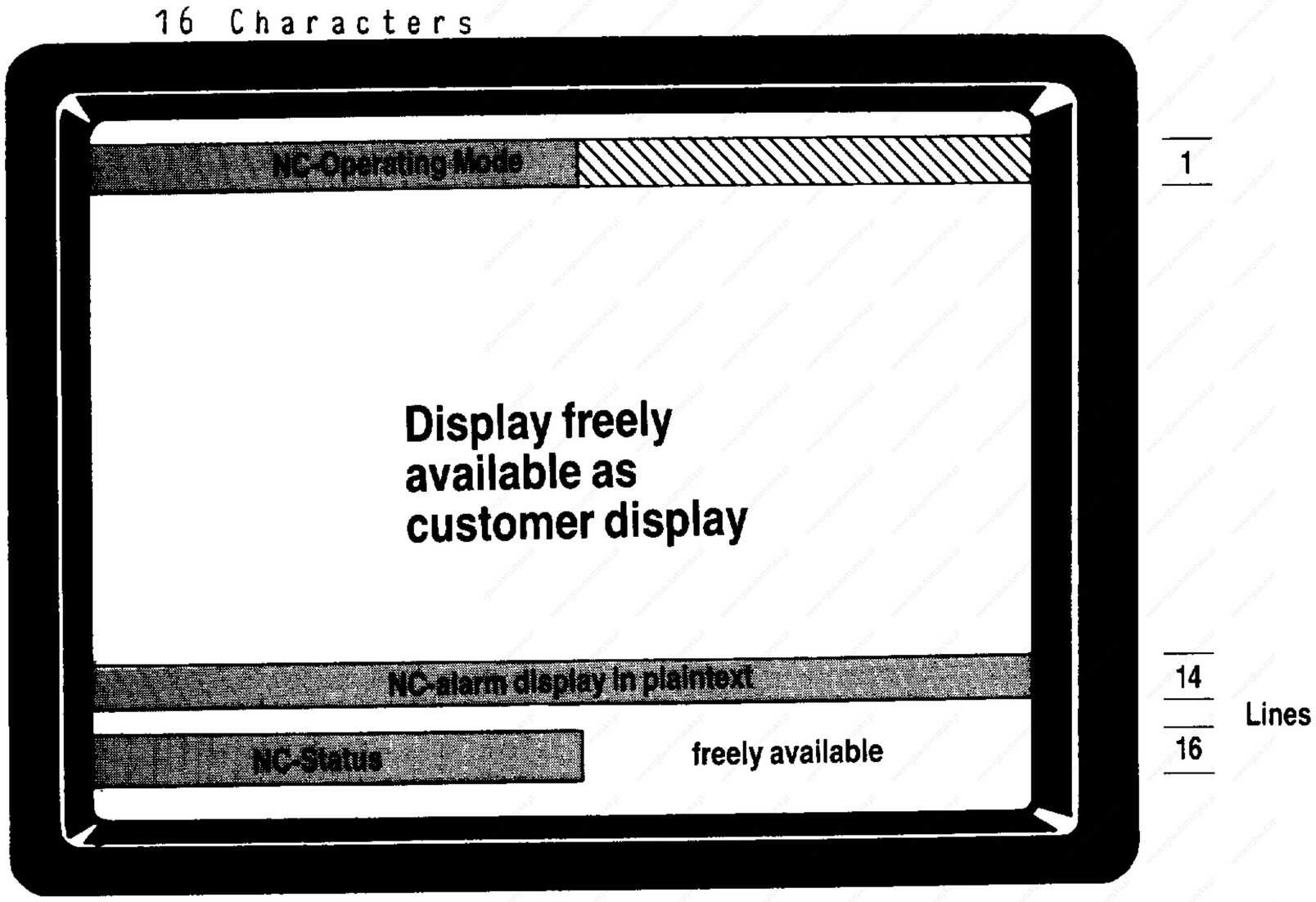
In Automatic mode, the "Block Search" function can be used to check whether the programs to be processed comply with the program format. The block search pointer is hereby set to the end of the part program. If an alarm is displayed, the block number of the last correct block is displayed (for selection and operation sequence, see Section 9.6.1).

9.8 Modifying the Information in Automatic mode via the Interface Control

NC-internal program processing can be modified with signals from the interface control, e.g. cancelling distances via measuring sensors or through the mirroring function (see the Interface Description).

9.9 Special Displays from the PLC

The CRT of the System 3 Controller can be used for the display of customized displays in all operating modes (customer display). The display formats and contents are defined by the machine tool manufacturer.



Reserved for insertion of PLC information in the NC display, e.g.
- machine-/PLC alarms
- number of the NC system connected to the operator panel (3TT)

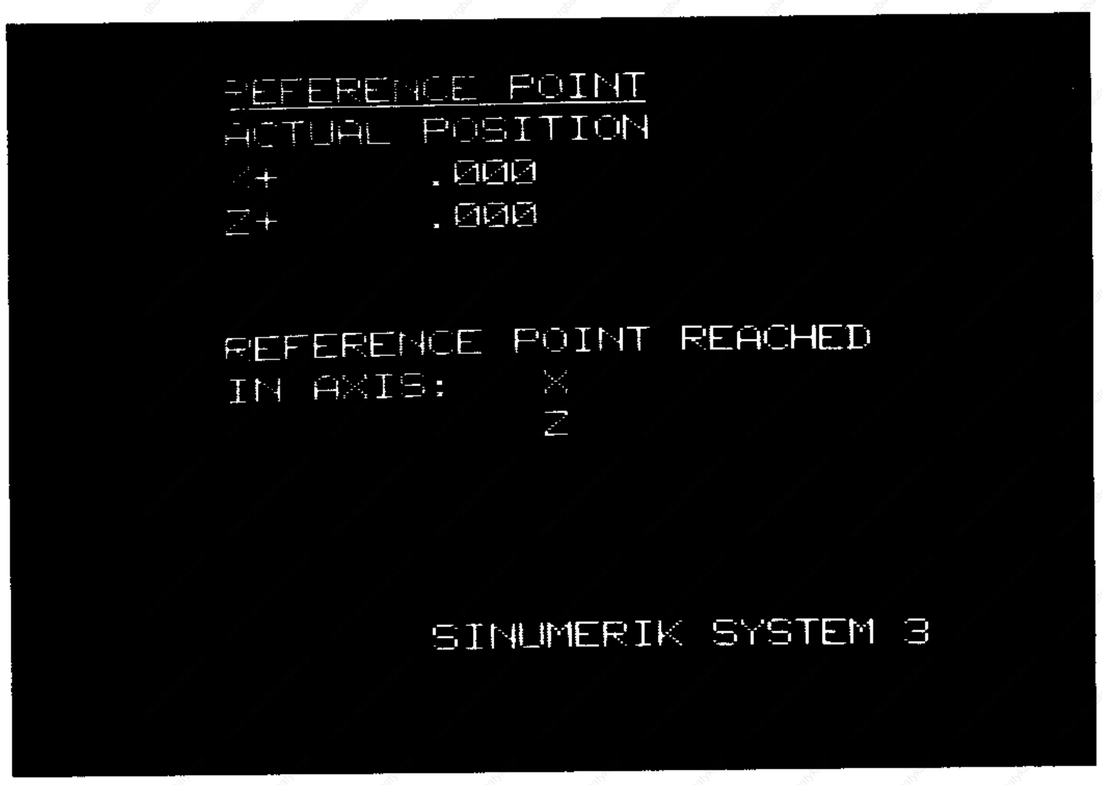
Reserved for insertion of NC information in the customer display, e.g.
- NC operating mode
- NC alarm display in plaintext
- NC status message.

10. Reference Point Approach

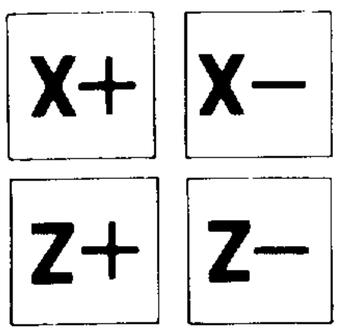
A reference point is defined for each axis of the machine, and serves as the starting point for the program.

This establishes a reference between the measuring system and the machine, and allows them to be synchronized at any time, even after the control has been switched off.

Reference point approach mode (selector switch No. 31)

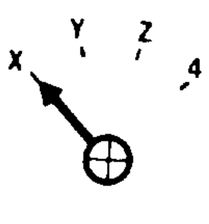


For the 3T

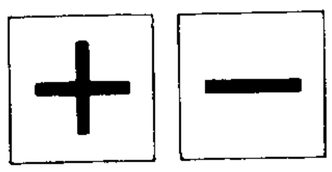


After pressing the appropriate direction key (No. 34), the axis moves to the reference point (modal direction).

For the 3M



The axis is selected with the axis selector switch (No. 47).



The axis moves to the reference point after the appropriate key (No. 34) has been pressed (modal direction).

The names of the axes that have been moved to the reference point are displayed.

The direction selected is checked before starting the traversing. (Wrong direction: operator input is rejected, i.e. there is no movement.)

The reference point approach must be executed sequentially for each axis.

When the reference point has been reached, the actual value counter is set to the value given as the reference point coordinate for the respective axis.

The feed hold and the feedrate/rapid override switch are active when the "Rapid override active" switch (No. 43) is activated.

The reference point approach speed is defined via machine data during start-up.

11. Operator Guidance with and without Graphics and Graphic Simulation

11.1 Introduction

- These Operating Instructions apply to SINUMERIK 3T with graphics and SINUMERIK 3M.

- The differences between the operation sequences with and without simulation detailed in the individual sections.

- The operation sequences are identical for colour and monochrome graphics systems or systems without graphics.

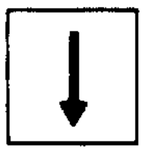
- Part program in "Automatic" mode can be programmed with operator prompting (without simulation)

11.1.1 Explanation of keys

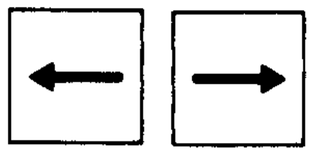
Confirmation of menu element
Acceptance of offered or input values

Continuation of menu (n-times NO returns from any position to the basic menu)

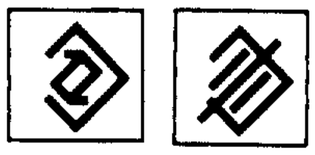
Switch back within the offered menu



Selection of next cycle or simulation display, confirmation and acceptance of total screen contents

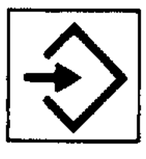


Positioning of cursor for program editing

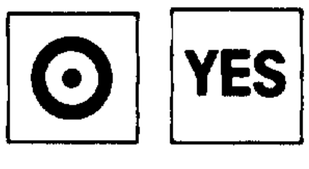


Correction of values which have been already transferred to the program by pressing YES

Deletion of data input in the input line by the operator



Only effective if input enabled



Start of simulation (after prompting)

Abort operator guidance and/or simulation
This causes a return to the basic menu of operator guidance.

11.1.2 Dialog Line

The dialog line is the last-but-one display line.

It offers:

- Remarks e.g. (SIMULATION AKTIV/SIMULATION ACTIVE)
- Requests e.g. (EINGABE MIT YES ODER NO/INPUT YES OR NO)
- Assistance for decision making, e.g.
  (FALSCHER SATZ WAEHLE AUT-PP/WRONG BLOCK,
  SELECT AUT-PP)

11.1.3 Data Input Line

The right-hand part of the last line serves as data input line. In this part of the CRT display, the data input via the keyboard appear.

11.1.4 Graphic Representation

For graphics simulation, the following basic types of representation are used:

Line graphics

All movements of the tool tip are represented in lines.

Rapid movements:	in a broken line	red
Feed movements :	continuous line	red
Threads:	dotted line	red

Optical cutting

The profile of the defined blank is shown in the selected view and coloured background. When the tool cuts into the workpiece, the coloured background is erased, in the shape of the tool. This optical stock removal is made along the programmed path as long as the tool has contact with the workpiece.

3T simulation

The simulation is represented in line graphics. Holes, tapped holes and cuts with structural steel shapes are represented by a single line.

3M simulation

A choice can be made from the following types of representation

- Line graphics for cutter centre path
- Optical cutting
- Optical cutting with additional representation of the cutter center path in the case of colour graphics

Parameter assignment in the machinging cycles is supported by static graphic display for 3T and 3M.

11.1.5 General Notes on Operator Guidance

The individual items in the operator guidance menus are selected via the "YES" and "NO" keys.
Addresses such as block address N etc. are automatically generated in the input line. If an input has already been made, the required figures are offered again. They can be confirmed with YES or overwritten by inputting a new value followed by YES.

The block numbers are generated in steps of 5.

Operator guidance is based on the NC program code in accordance with DIN 66025. The DIN code corresponding to the menu selection is shown in the input line and is transferred into the program memory and to the display by pressing the YES key.

The graphics functions for simulation and cycle parameter assignment with static displays are integrated into operator guidance and can only be used if operator guidance is activated.

11.1.6 Activating Operator Guidance

1. Setting data 25, bit 1 = 1
2. Set mode selection switch to MDE-PP or AUT.

11.2 Part Program Input with Operator Guidance

11.2.1 Input of a New Part Program

1. Key-operated switch released.
   (depends on machine datum)

2. Select "operator guidance" for part program or subroutine in the menu.

3. Input part program or subroutine number.

4. With operator guidance it is now possible to input any desired work cycle, geometrical data, technological values, special functions, such as T, M, H functions, etc.

   w i t h g r a p h i c s

5. The prepared program section may be simulated on the display after each block (see Section 11.4).

Exception:
For simulating blocks with contour programming (insertion of radi and chamfers), the controller requires the following block in order to correctly insert the radius or the chamfer between the two blocks.

When using operator guidance, this is done automatically and the control does not request the simulation at the end of a block with a contour element. The following block must be input first.

If, during free input in DIN code without operator guidance, the following block for a contour element is not yet defined, and the simulation is started, the control outputs an error message.

11.2.2 Termination of a New Part Program

1. Select "SONDERFUNKTIONEN/SPECIAL FUNCTIONS" menu.

2. Proceed to "PROGRAMMENDE/PROGRAM END" menu by pressing "NO".

3. Select "PROGRAMMENDE/PROGRAM END" menu.

4. Terminate the part program.

   w i t h o u t g r a p h i c s

5. Return to "FREIE EINGABE MIT TASTE INPUT/FREE INPUT USING THE KEY INPUT" basic menu.

   w i t h g r a p h i c s

5. The "PROGRAMMSIMULATION/PROGRAM SIMULATION" menu is offered.

6. Now continue simulation parameter assignment (Section 11.4).

11.2.3 Inputs into an Existing Part Program

1. Select operator guidance for part program or subroutine in the menu.

2. Select part program or subroutine number.

3. Select the "PROGRAMMEINGABE ODER KORREKTUR/PROGRAM INPUT OR COMPENSATION" menu.

4. Position cursor.

5. Select input with or without operator guidance.

11.2.4 Deletion of a Part Program

(see also Operating Instruction, Part 3)

1. Set mode selector switch to MDI-PP:

2. Press the "Anzeige umschalten/change display" key once.

3. Input the part program number to be deleted (e.g. 90 1234 or L123) and press the "Cancel" key.

11.2.5 Correction of Stored Part Programs

Correction is possible in the "FREIE EINGABE MIT TASTE INPUT/ FREE INPUT USING THE INPUT KEY" basic menu without operator guidance.
(see Operating Instructions System 3, Part 3)

11.2.6 Input of Parameters for Machining Cycles with Graphics Support

Static graphics displays support parameter assignment for the machining cycles stored in the control if the graphics option is available.

Select the desired cycle from the overview display by pressing the "YES", "NO" and "PAGE" keys.

The control then requests that all parameters be entered (e.g. STARTPUNKT X?/STARTING POINT X?).

Preset parameters can be accepted by pressing "YES".

By means of "Zahlenwert-YES/numeric value-YES", a new value is input. The next display appears when "Page" is pressed.

The parameter to be input is displayed in inverse video.

In case of erroneous inputs, return to the parameter by using the back-space key and input the correct parameter value.

After the input or, confirmation of all parameters required for the desired cycle, the "PARAMETERWERTE RICHTIG?/PARAMETER VALUES CORRECT? question appears.

If the answer is "No", the parameters are again offered and may be corrected.

If the parameter values of the selected cycle are not be taken into the program, answer the "PARAMETER ABSPEICHERN?/STORE PARAMETERS? question with "NO".

This function permits the control to be used as "graphics editing support" for correcting R parameters in the stored program.

11.2.7 Correction of Parameters for Machinging Cycles

1. Delete existing cycle call in the "FREIE EINGABE MIT TASTE INPUT/ FREE INPUT USING THE INPUT KEY" menu including the defined parameters in the part program (see Operating Instructions, Part 3).

2. Select operator guidance in the menu for part program or subroutine.

3. Choose part program or subroutine.

4. Select "PROGRAMMEINGABE ODER KORREKTUR/PROGRAM INPUT OR CORRECTION" menu.

5. Position cursor.

6. Select menu for cycle input and input the corresponding new values (see 11.1.6).

11.2.7.1 Correction of Individual Parameters for Machinging Cycles

1. Select operator guidance in the menu for part program or subroutine.

2. Choose part program or subroutine.

3. Select the "PROGRAMMEINGABE ODER KORREKTUR/PROGRAM INPUT OR CORRECTION" menu.

4. Position the cursor on the parameter to be changed.

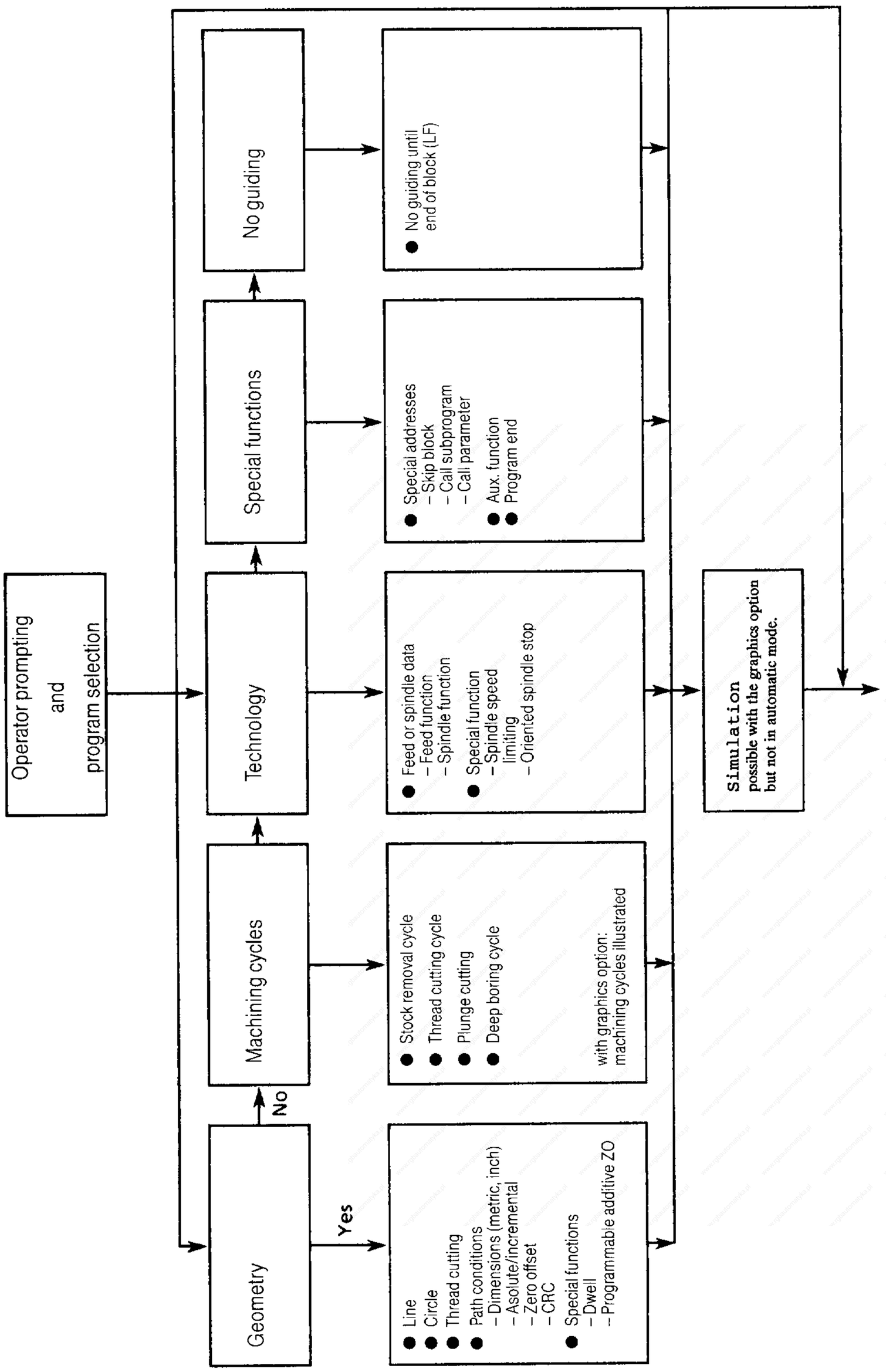
5. Select menu for cycle input and the desired machining cycle.

6. Select the desired parameter in the cycle display.

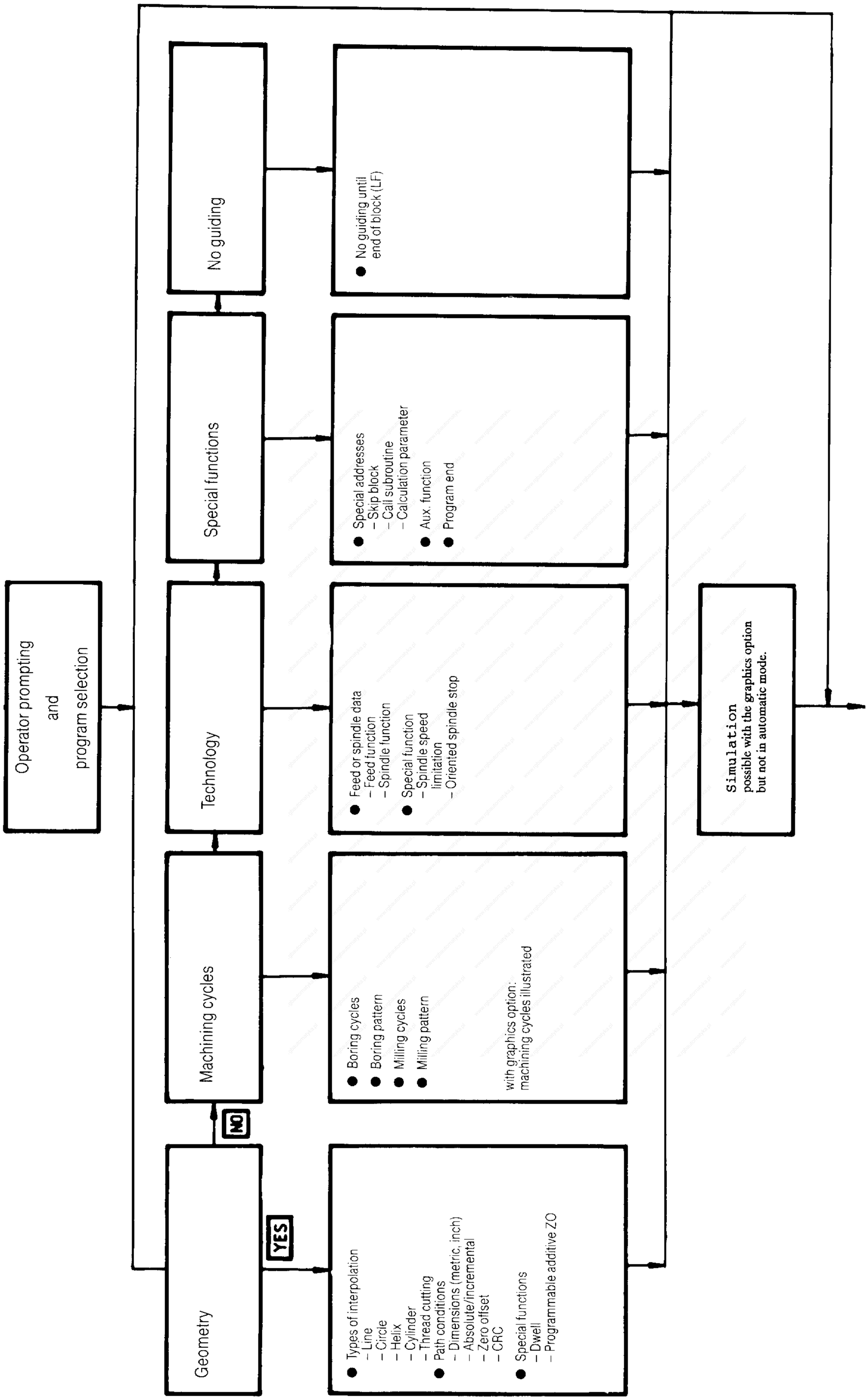
7. Input new value and transfer by pressing the "Edit" key (this means that the old value is corrected).

8. Respond with "NO" to" PARAMETER ABSPEICHERN /STORE PARAMETERS".

11.2.8 Schematic Overview of Operator Guidance (3T)



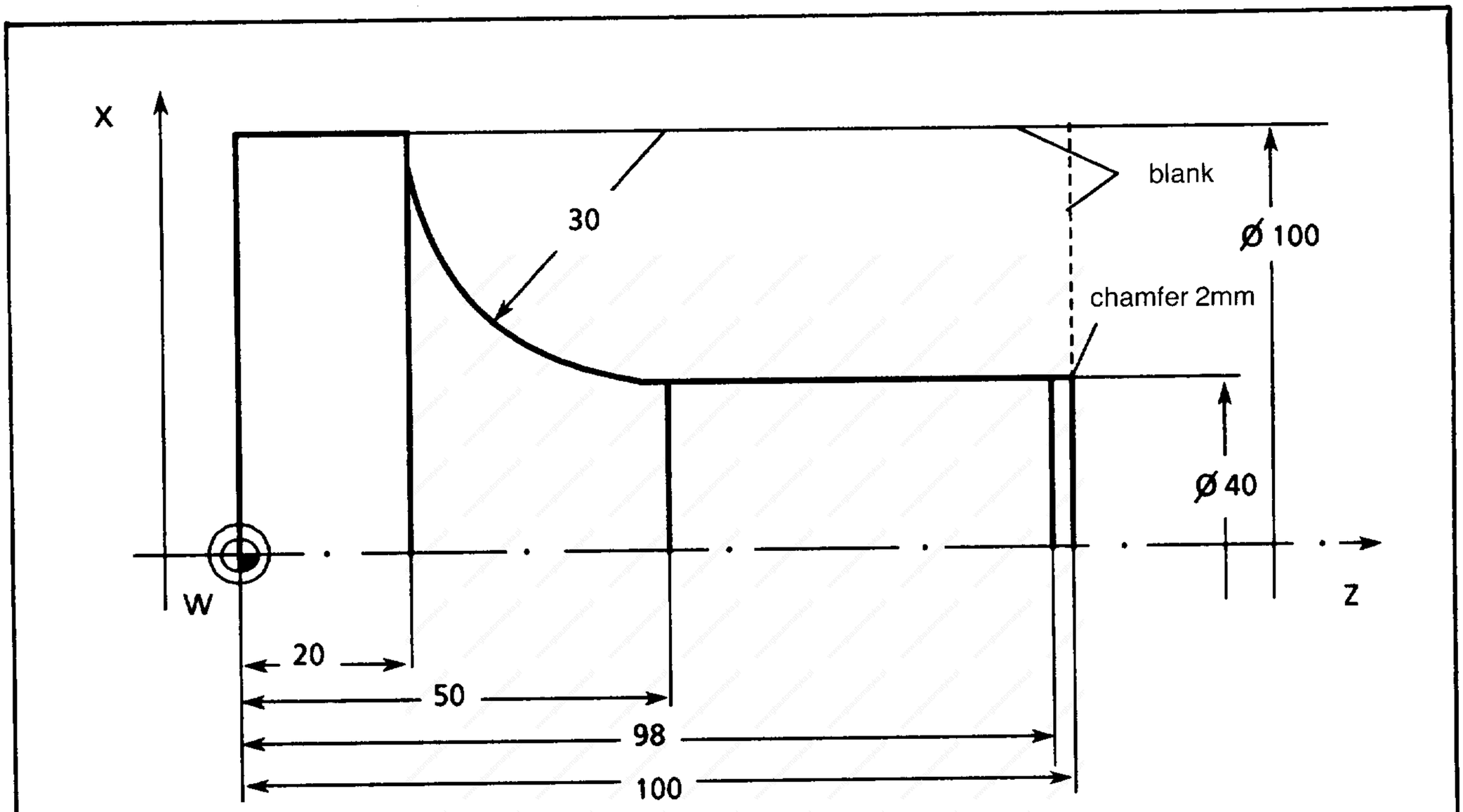
11.2.9 Schematic Overview of Operator Guidance (3M)



11.3 Operation Sequence for Program Example

11.3.1 Program Example 1 (3T)

Prepare the following parts program using operator guidance with and without graphics.



Input without graphics support

```
%1234
N5  G95  F1  S1000  M03
N10  R20  11  R22  3  R21  2  R16  1  R17  1  R183
N15  R10  100  R11  20  R12  40  R13  50
N20  R23  0  R24  30  R14  100  R15  110
N30  L9401
N35  X0  G00
N40  Z100
N45  X40  G01  B-2
N50  Z50
N55  X100  Z20  G02  B30
N60  Z0  G01
N65  M30
```

Input with graphics support

```
%1234
N5  G95  F1  S1000  M03
N10  R20  11  R22  3  R21  2  R16  1
N15  R17  1  R18  3  R10  100  R11  20
N20  R12  40  R13  50  R14  100  R15  110
N25  R23  0  R24  30
N30  L9401
N35  X0  G00
N40  Z100
N45  X40  G01  B-2
N50  Z50
N55  X100  Z20  G02  B30
N60  Z0  G01
N65  M30
```

11.3.1.1 Operation Sequence for Program Example 1 (3T)

Activate operator guidance (see 11.1.5).

Key-operated switch released (No. 38)

dependent on machine datum

NO Select operator guidance for part program.

1 **2** **3** **4** Input program number

YES Open program

11.3.1.2 Input Block N5

N5

YES Store offered block number for the part program memory

with

NO and **YES** Select "TECHNOLOGIE/TECHNOLOGY" menu

2x

YES Select feedrate and spindle value input

G95

YES Store offered function feedrate per revolution

F

1 Input feedrate

YES Store feedrate

S
1 **0** **0** **0** Input spindle speed

YES Store spindle speed
M03

YES Store offered direction of rotation (M03 in CW)

*
YES Terminate block

with the GRAPHICS option, it is possible to simulate the input block.

NO No simulation

or

YES Simulation (see Section 11.3)

11.3.1.3 Input Block N10-N30

(Parameter input for cutting cycle with graphic support).

N10

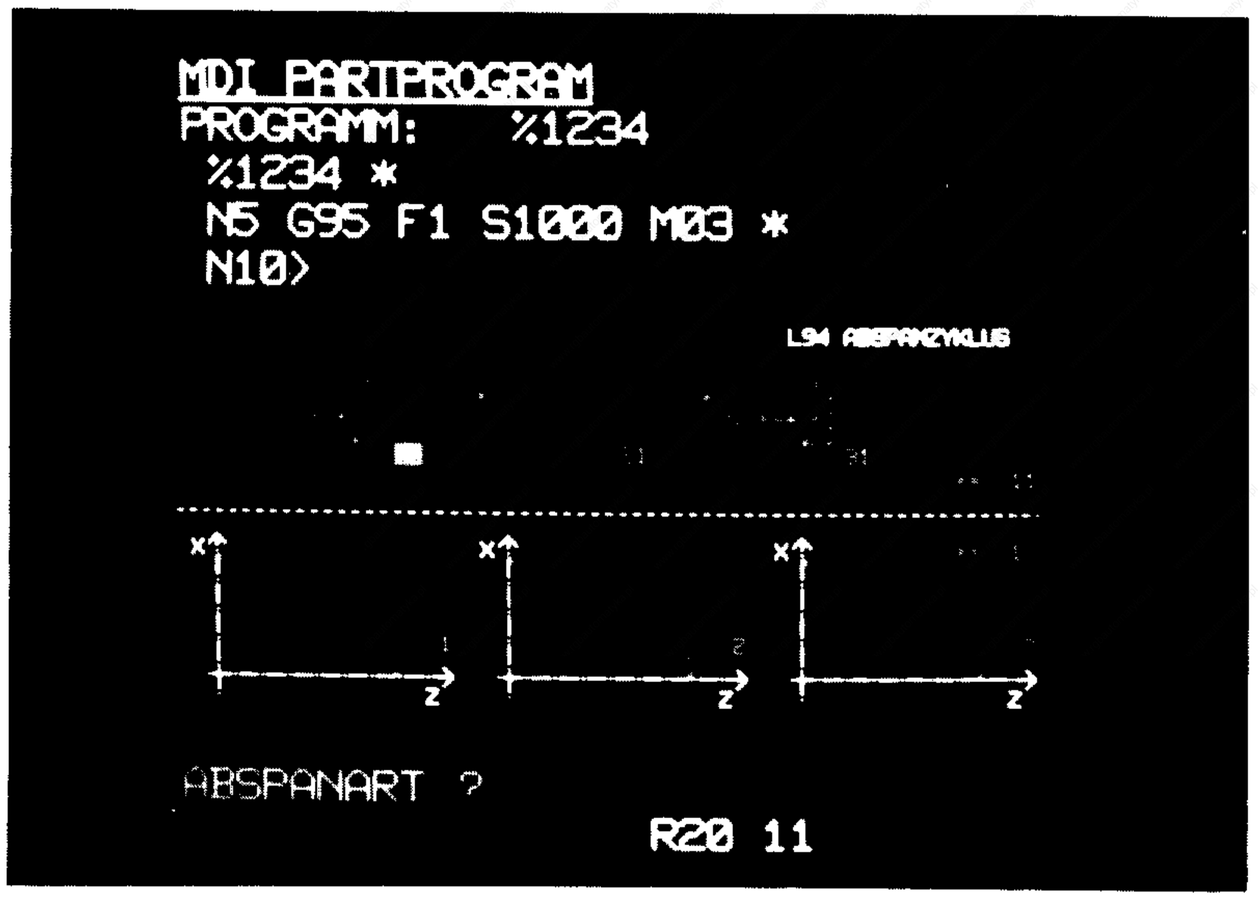
YES Store offered block number

with

NO and **YES** Select "ARBEITSZYKLEN/WORK CYCLES" menu.

YES Select cutting cycle L94

with the GRAPHICS option, the following display appears for the cycle parameters



without the GRAPHICS option, see Section 11.2.10.3

R2011

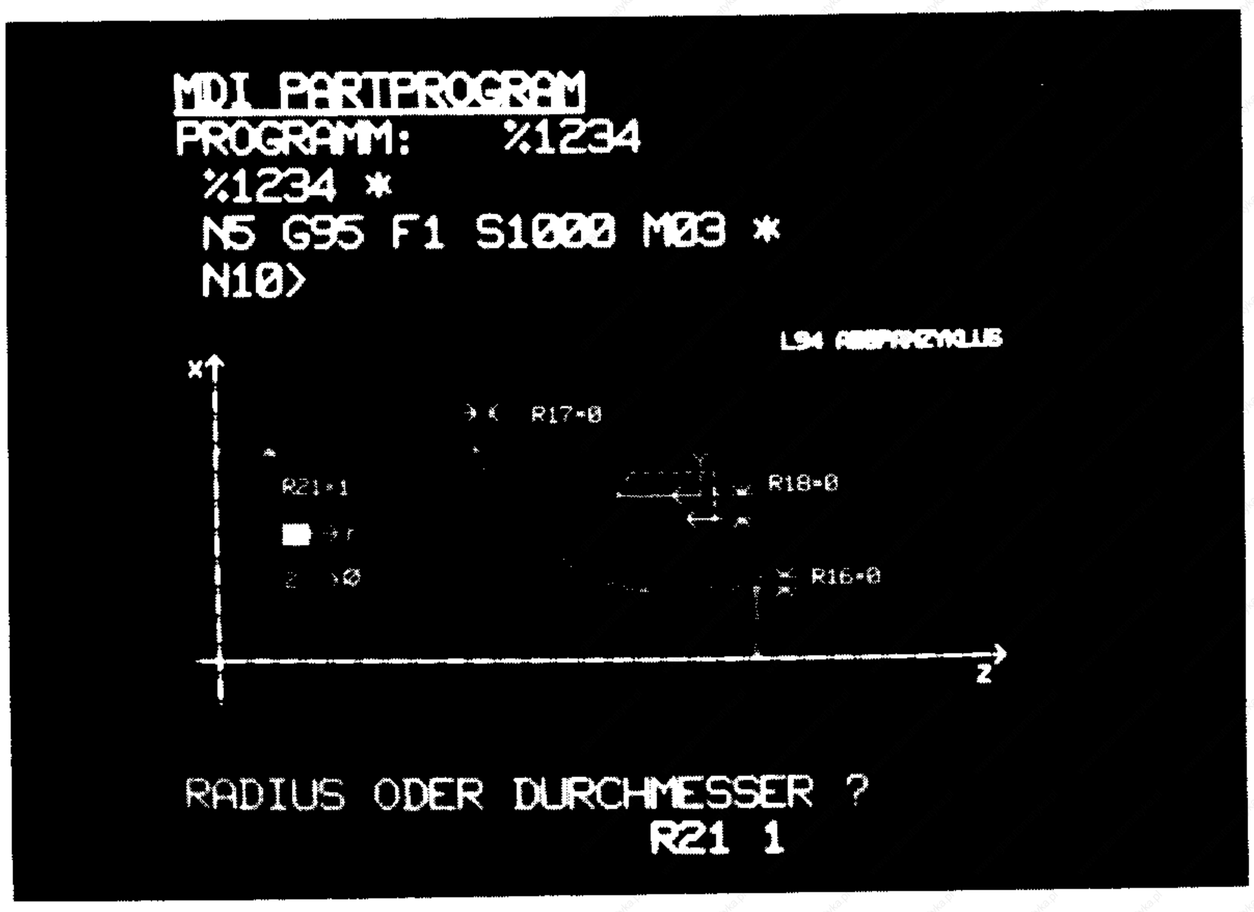
YES Confirm cutting process od longitudinal

R22 3

with

NO and **YES** Confirm cutting element circular arc towards id

The following display appears:



R21 1

NO

No radius input

R21 2

YES

Select diameter input for X values and confirm

R16 0

1

Input finishing allowance of 1 mm in X direction and confirm with **YES**

R17 0

1

Input finishing allowance of 1 mm in Z direction and confirm with **YES**

R18 0

3

Input cutting depth of 3 mm and conform with **YES**

The following display appears:

R10 0

1 **0** **0**

Input starting point X 100 mm Ø and confirm with **YES**

R11 0

2 0

Input starting point Z 20 mm and confirm with YES

R12 0

4 0

Input end point X 40 mm Ø

R12 40

and confirm

YES

R13 0

5 0

Input end point Z 50 mm and confirm with YES

R14 0

1 0 0

Input return point in X 100 mm Ø and confirm with YES

R15 0

1 1 0

Input return point in Z 110 mm and confirm with YES

R23 0

Confirm X circular parameter offered

YES

R24 0

3 0

Input Z circular parameter 30 mm and confirm with YES

with

YES

Proceed if all values have been correctly input and confirmed

with

YES

Store confirmed parameters; this is done in different blocks

N15

YES If the block number is desired during storage of parameters, then

N20 to N30

YES answer with **YES** each time.

The menu display then returns.

L9401

YES Stored offered cycle number

* and terminate block

YES

with

NO No simulation

or with

YES Simulation (see Section 11.3)

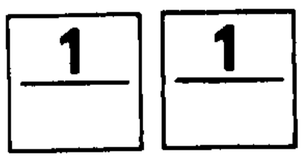
<u>End of the cycle input with graphic support (continue with 11.3.1.5)</u>

11.3.1.4. Input of Parameters for Cutting Cycle without Graphics Support (up to block N30)

with

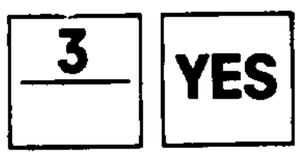
Select new cycle input

R20



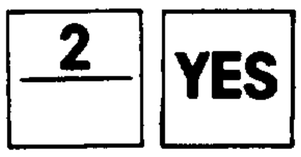
Input cutting process odd longitudinal and store with

R22



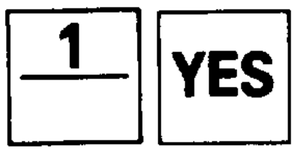
Input cutting element circular arc and store

R21



Determine diameter input for X values and store.

R16

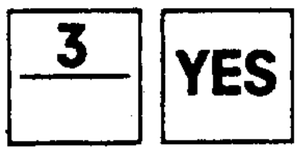


Input finishing allowance of 1 mm in X direction and store

R17

Input finishing allowance of 1 mm in Z direction and store

R18



Input cutting depth of 3 mm and store

N15

Accept offered block number and then proceed with parameter input

R10

1 0 0 Input starting point in X Ø 100 mm and store

R11

2 0 Input starting point in Z 20 mm and store with YES

R12

4 0 Input end point in X Ø 40 mm and store with YES

R13

5 0 Input end point in Z 50 mm and store with YES

N20

YES Accept offered block number and proceed with parameter input

with

YES Select circular element cutting

R23

0 YES Input circular parameter in X = 0 (interpolation parameter I) and store

R24

3 0 Input circular parameter in Z = 30 mm (interpolation parameter K) and store with YES

R14

1 0 0 Input return point in X Ø 100 mm and store with YES

R15

1 1 0 Input return point in Z 110 mm and store with YES

N25

NO

3 0 Reject the offered block number

Input new block number and store with YES

L9401

Accept offered cycle call

YES

*

YES Terminate block

11.3.1.5 Input Block N35

N35

YES Accept offered block number

YES Select geometrical input

YES Select line programming

X

0 Input starting point for X and store with **YES**

Z

NO

GOO

YES Store

2x

NO

* Terminate block

YES

with the GRAPHICS option, it is possible to simulate the block input.

NO No simulation

or

YES Simulation (see Section 11.3)

11.3.1.6 Input Block N40

N40

YES Input offered block number

2x

YES Select geometry and linear interpolation

X

NO

Z

1 **0** **0** Input end point Z 100 and store with **YES**

with

NO

and

YES Select block termination

*

YES Block termination

with the GRAPHICS option, it is possible to simulate the block input.

NO No simulation

or

YES Simulation (see Section 11.3)

11.3.1.7 Input Block N45

N45

YES Store offered block number

with

YES Select geometry and linear interpolation

X

4 **0** Input X value and store with **YES**

Z

NO No Z value

G00

NO

G01

YES Select linear interpolation with feed rate

YES Select contour programming

B

NO Do not insert radius but

B

— **2** insert chamfer

*

YES Terminate block and store

11.3.1.8 Input Block N50

N50

YES Store offered block number

with

YES Select geometry and linear interpolation

X

NO No X value

Z

5 **0** Input Z value and store with **YES**

with

NO Preset the block termination function

*

YES and store block end

with the GRAPHICS option, it is possible to simulate the block input.

NO No simulation

*

YES Simulation (see Section 11.3)

11.3.1.9 Input Block N55

N55

YES

Store offered block number

with

YES

Select geometry and circular programming

and

NO

Select

X

1 **0** **0**

Input circular end point in X and store with **YES**

Z

2 **0**

Input circular end point in Z and store with **YES**

G02

YES

Store direction of rotation CW

B

3 **0**

Input circular radius and store with **YES**

G09

NO

No exact stop

*

YES

Terminate block

with the GRAPHICS option, it is possible to simulate the block input.

NO

No simulation

or

YES

Simulation (see Section 11.3)

11.3.1.10 Input Block N60

N60

YES Store offered block number

with

YES Select geometry and linear programming

and

NO

Z

0

Input Z value and store with **YES**

G00

NO

G01

YES Select linear interpolation with feedrate and store

with

NO Select the block termination function

and

YES

*

YES Terminate block

with the GRAPHICS option, it is possible to simulate the block input.

NO No simulation

or

YES Simulation (see Section 11.3).

11.3.1.11 Input Block N65

N65

YES Store offered block

with

NO

and

YES Select "SONDERFUNKTIONEN/SPECIAL FUNCTIONS" menu

with

NO

and

YES Select program end

M30

YES Store program end

*

YES Terminate the last program block

with the GRAPHICS option, the basic menu for free input with the Input key returns.

or

with the GRAPHICS option, it is possible to simulate the input program.

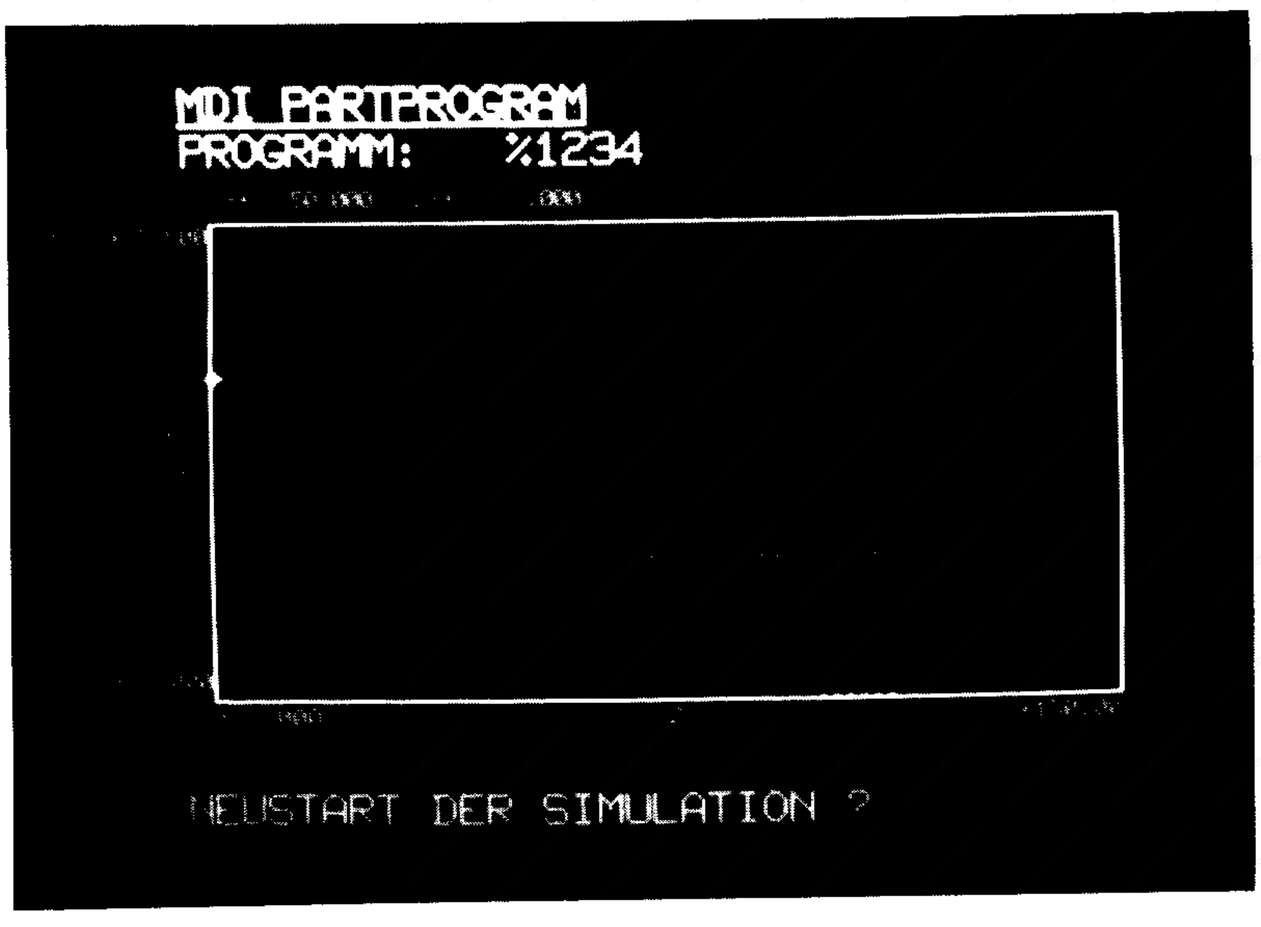
with **NO** Return to the "FREIE EINGABE MIT DER TASTE INPUT/ FREE INPUT USING THE INPUT KEY" basic menu

or with **YES** proceed with the menu for program simulation

11.3.1.12 Simulation of Program Example 1 (3T)

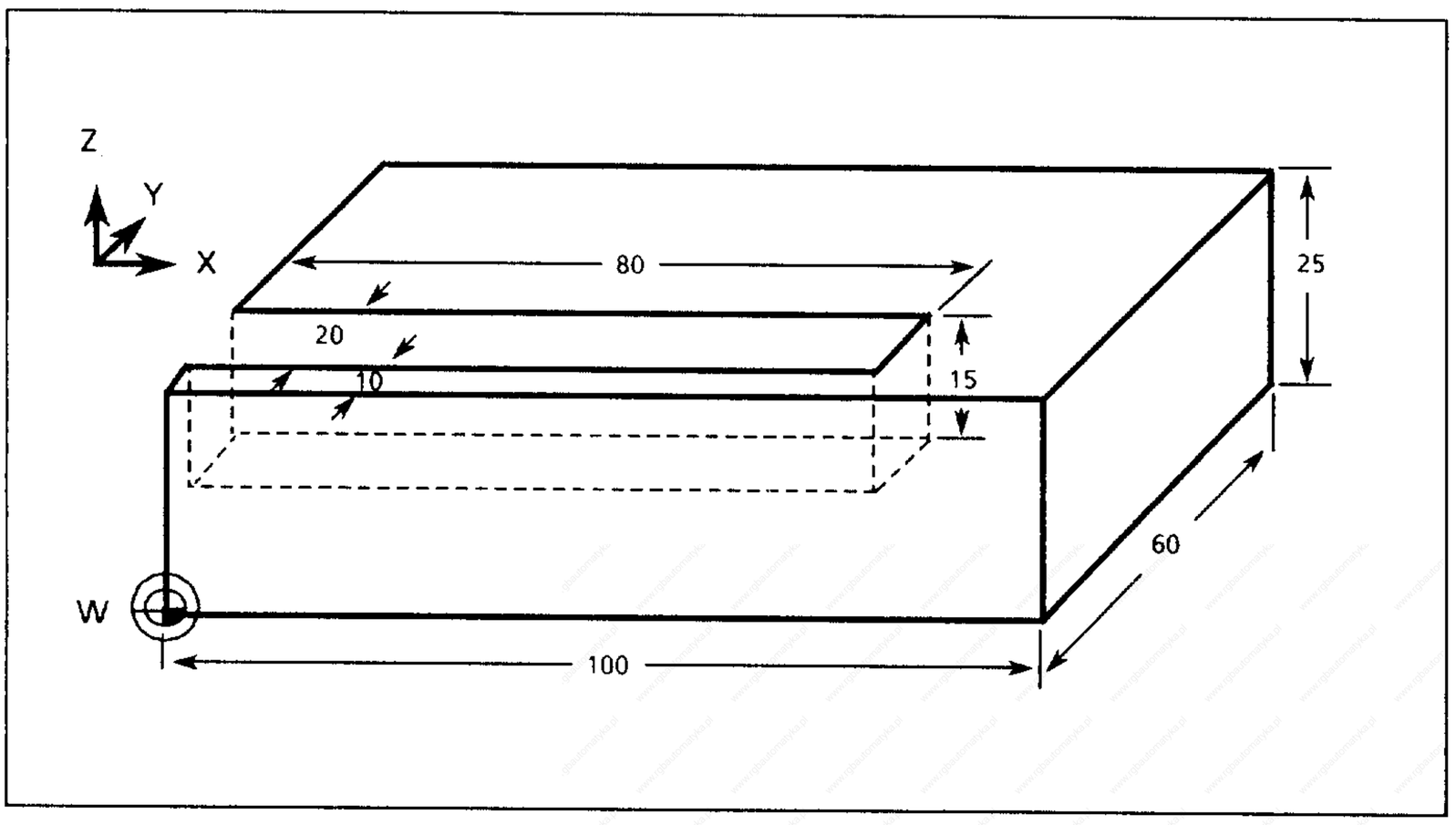
For the program example %1234, the simulation parameters have to be assigned the following values via operator guidance (see Section 11.3)

R10	Simulation range X-MIN Ø	mm
R11	Simulation range Z-MIN Ø	mm
R12	Simulation range X-MAX Ø	150 mm
R13	Simulation range Z-MAX	150 mm
R14	Workpiece diameter X Ø	100 mm
R15	Workpiece diameter Z	100 mm
R21	Diameter input	= 2



11.3.2 Program Example 2 (3T)

Create the following part program by means of operator guidance.



Cutter radius 8 mm

W = Workpiece zero

Part program created with operator guidance

```
%1234
N5 G94 F1000 S1000 M03
N10 X-20 Y20 Z30 G00
N15 D01 G41 X-10 Y10
N20 Z20 G01
N25 X80
N30 Y30
N35 X-10
N40 G40 X-15 Y20 G00
N45 Z30
N50 M30
```

11.3.2.1 Operation Sequence for Program Example 2 (3M)

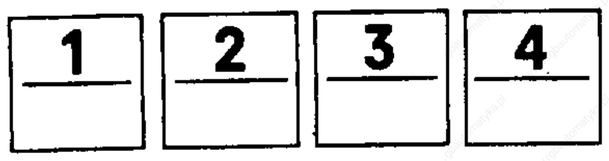
Activate operator prompting (see Section 11.1.6).

Release Key-operated switch (No. 38)

dependent on machine datum

Select part program

%

e.g. 

Input program number

Open program

11.3.2.2 Input Block N5

N5

YES Store offered block number for the part program memory

with

NO + **YES** Select "TECHNOLOGIE/TECHNOLOGY" menu

ZX

YES Select feedrate and spindle value input

G95

NO Reject offered function (feedrate per revolution)

G94

YES Accept offered function (linear feedrate)

F

1 **0** **0** **0** Input feedrate and store with **YES**

S

1 **0** **0** **0** Input spindle speed and store with **YES**

M03

YES Store offered direction of rotation

*

YES Terminate block

with the GRAPHICS option, it is possible to simulate the block input.

NO No simulation

or

YES Simulation (see Section 11.3)

11.3.2.3 Input Block N10

N10

YES — Store offered block number

with

YES — Select "GEOMETRIE/GEOMETRY" menu
then select linear interpolation via the
"INTERPOLATIONSART/TYPE OF INTERPOLATION" menu.

YES — Select cartesian coordinates

X

— 2 0 — Input end point X-20 and store with YES

Y

2 0 — Input end point X20 and store with YES

Z

3 0 — Input end point Z30 and store with YES

4. — No endpoint for 4th axis

NO

G00 — Select offered preparatory function.

YES

with

NO — Terminate block

and

YES

With GRAPHICS option, it is possible to simulate the block input.

NO — No simulation

or

YES — Simulation (see Section 11.3)

11.3.2.4 Input Block N15

N15

YES — Store offered block number

YES — Select "GEOMETRIE/GEOMETRY" menu

with

NO — Select "VORBEREITENDE FUNKTIONEN/PREPARATORY FUNCTIONS"

and

YES

with

NO + YES — Select "CUTTER RADIUS COMPENSATION" and "CRC selection"

D

0 1 — Input tool offset number and store with YES

G41

YES — Store offered function

with

NO — return to basic menu

with

YES — Select "GEOMETRIE/GEOMETRY" menu and then linear interpolation via the "INTERPOLATIONSART/TYPE OF INTERPOLATION" menu

YES — Select cartesian coordinates

X

[−] [1] [0] Input endpoint X-10 and store with

Y

[1] [0] Input endpoint Y10 and file with [YES]

Z

[NO] No endpoint Z axis

4.

[NO] No endpoint 4th axis

with

[NO]

and

[YES] terminate block

With GRAPHICS option, it is possible to simulate the block input.

[NO] No simulation

or

[YES] Simulation (see Section 11.4.3.)

11.3.2.5 Input Block N20

N20

YES Store offered block number

with

YES Select "GEOMETRIE/GEOMETRY" menu and then linear interpolation via the "INTERPOLATIONSART/TYPE OF INTERPOLATION" menu

YES

Select cartesian coordinates

2X

NO Page through to endpoint Z

Z

1 **0** Input endpoint Z10 and store with **YES**

4.

NO No endpoint 4th axis

G00

NO Reject offered preparatory function

G01

YES Store offered preparatory function

with

NO + **YES** Terminate block

With GRAPHICS option, it is possible to simulate the block input.

NO No simulation

or

YES Simulation (see Section 11.4.3).

11.3.2.6 Input Block N25

N25

YES — Store offered block number

with

YES — Select "GEOMETRIE/GEOMETRY" menu and linear interpolation via the "INTERPOLATIONSART/TYPE OF INTERPOLATION" menu.

YES — Select cartesian coordinates

X

8 **0** — Input endpoint X80 and store with **YES**

with

NO + **YES** — Terminate block

With GRAPHICS option, it is possible to simulate the block input.

NO — No simulation

or

YES — Simulation (see Section 11.4.3)

11.3.2.7 Input Block N30

N30

YES — Store offered block number

with

YES — Select "GEOMETRIE/GEOMETRY" menu and linear interpolation via the "INTERPOLATIONSART/TYPE OF INTERPOLATION" menu.

YES — Select cartesian coordinates

X

NO — Page through to endpoint Y

Y

3 **0** — Input endpoint Y30 and store with **YES**

with

NO — Terminate block

and

YES

With the GRAPHICS option, it is possible to simulate the block input.

NO — No simulation

or

YES — Simulation (see Section 11.4.3)

11.3.2.8 Input Block N35

N35

YES Store offered block number

with

YES Select "GEOMETRIE/GEOMETRY" menu and linear interpolation via the "INTERPOLATIONSART/TYPE OF INTERPOLATION" menu

YES Select cartesian coordinates

X

— **1** **0** Inpunt endpoint X-10 and store with **YES**

with

NO + **YES** Terminate block

With the GRAPHICS option, it is possible to simulate the block input.

NO No simulation

or

YES Simulation (see Section 11.4.3)

11.3.2.9 Input Block N40

N40

YES — Store offered block number

with

YES + NO — Select "GEOMETRIE/GEOMETRY" menu and then "VORBEREITENDE FUNKTIONEN/ PREPARATORY FUNCTIONS" menu

with

NO + YES — Select cutter radius compensation deselection

G40

YES — Store offered FRK-Abwahl/cutter radius compensation deselection function

with

NO — Return to basic menu

with

YES

YES — Select GEOMETRIE/GEOMETRY" and then linear interpolation via the "INTERPOLATIONSART/TYPE OF INTERPOLATION" menu

X

— 1 5 — Select cartesian coordinates

Y

2 0 — Input endpoint X-15 and store with

Z

NO — Input endpoint Y20 and store with

4.

YES — No endpoint Z axix

No endpoint 4 th axis

GOO

YES Accept offered preparatory function

with

NO

and

YES Terminate block

With the GRAPHICS option, it is possible to simulate the block input.

NO No simulation

or

YES Simulation (see Section 11.4.3)

11.3.2.10 Input Block N45

N45

YES Store offered block number

with

YES Select linear interpolation with the "GEOMETRIE/ GEOMETRY" menu and then linear interpolation via the "INTERPOLATIONSART/TYPE OF INTERPOLATION" menu

YES Select cartesian coordinates

2*

NO Page through to endpoint Z

Z

3 **0** Input endpoint Z30 and store with **YES**

with

NO + **YES** Terminate block

With the GRAPHICS option, it is possible to simulate the block input.

NO No simulation

with

YES Simulation (see 11.4.3)

11.3.2.11 Input Block N50

N50

YES — Store offered block number

with

NO + **YES** — Select "SONDERFUNKTIONEN/SPECIAL FUNCTIONS" menu and page through to the "PROGRAMMENDE/PROGRAM END" menu

YES — Select "PROGRAMMENDE/PROGRAM END" menu

M30

YES — Store offered program end

*

YES — Terminate block

With the GRAPHICS option, it is possible to simulate the input program.

with

NO — Return to basic "FREIE EINGABE MIT TASTATUR/FREE INPUT VIA KEYBOARD" menu

or

with

YES — Continue with the "PROGRAMMSIMULATION/PROGRAM SIMULATION" menu (see Section 11.3.2.12)

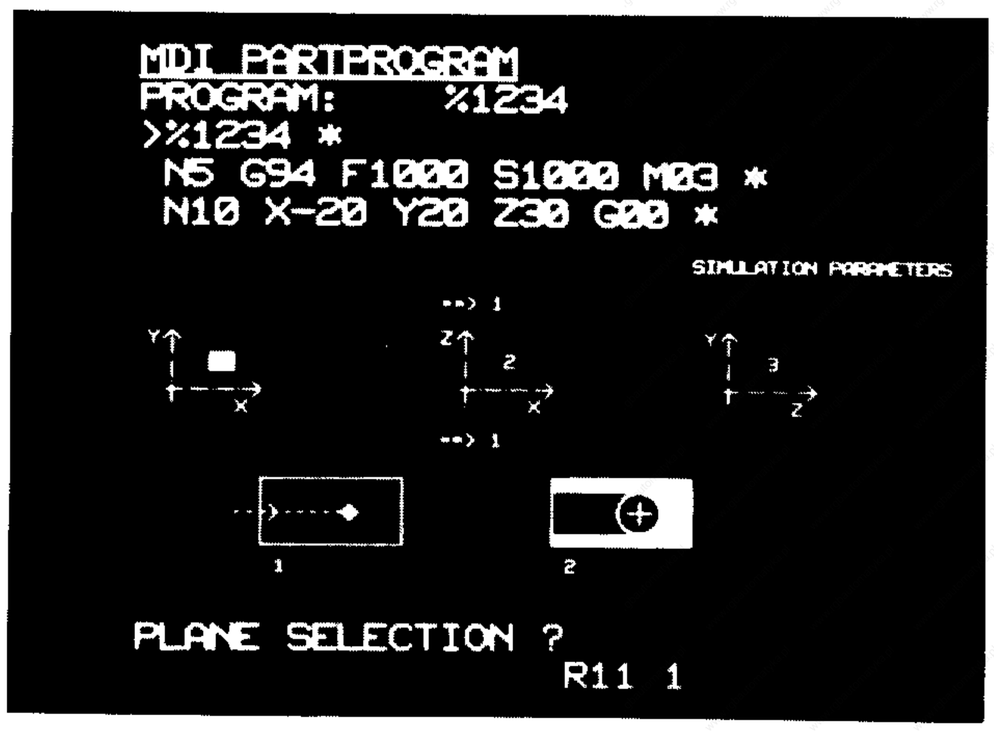
11.3.2.12 Simulation of Program Example 2 (3M) 3/4

For the simulation of the program example %1234 for the 3M, the simulation parameters must be defaulted using operator guidance. In addition, the values for cutter radius and length must be stored in the tool offset memory DO1.

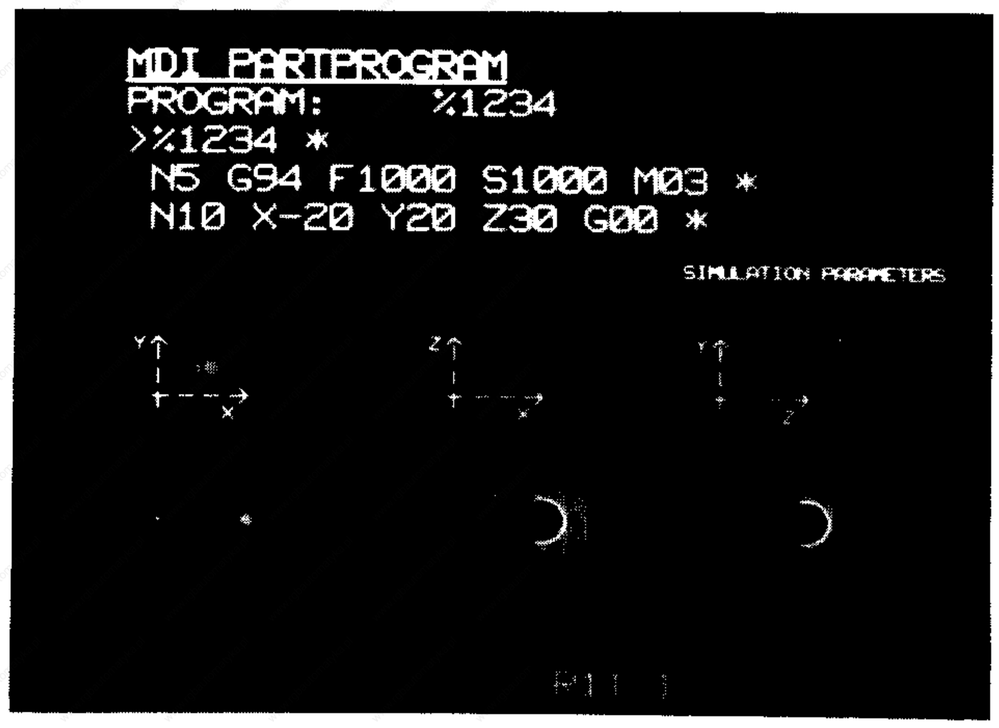
Select the "SIMULATIONSBEREICH DEFINIEREN/DEFINE SIMULATION RANGE" menu with **YES**

The following display appears

for monochrome graphics



for colour graphics



with

NO Select plane in which the programmed part is to be simulated.

and

YES in this example, plane X-Y

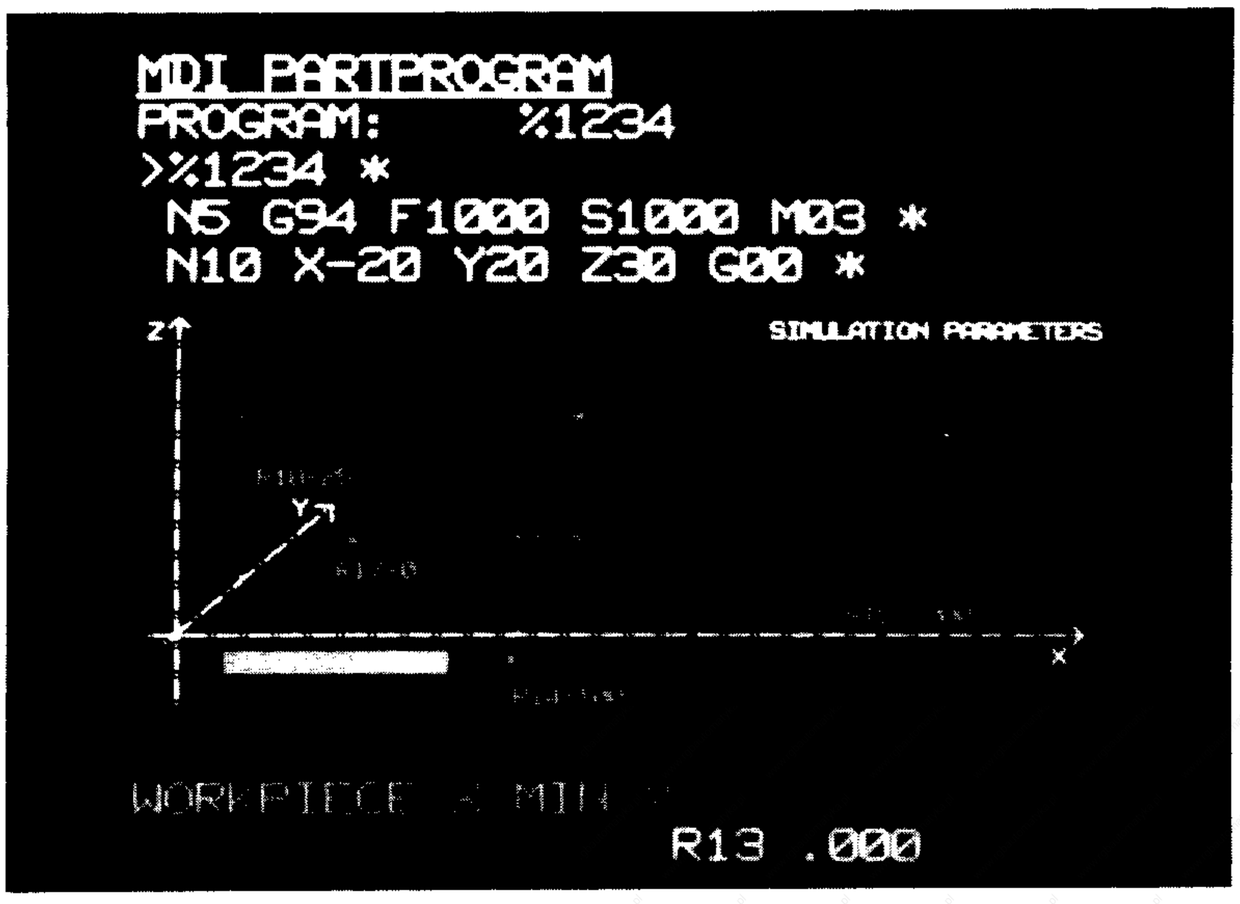
with

NO Type of graphic representation, e.g. select optical cutting

and

YES

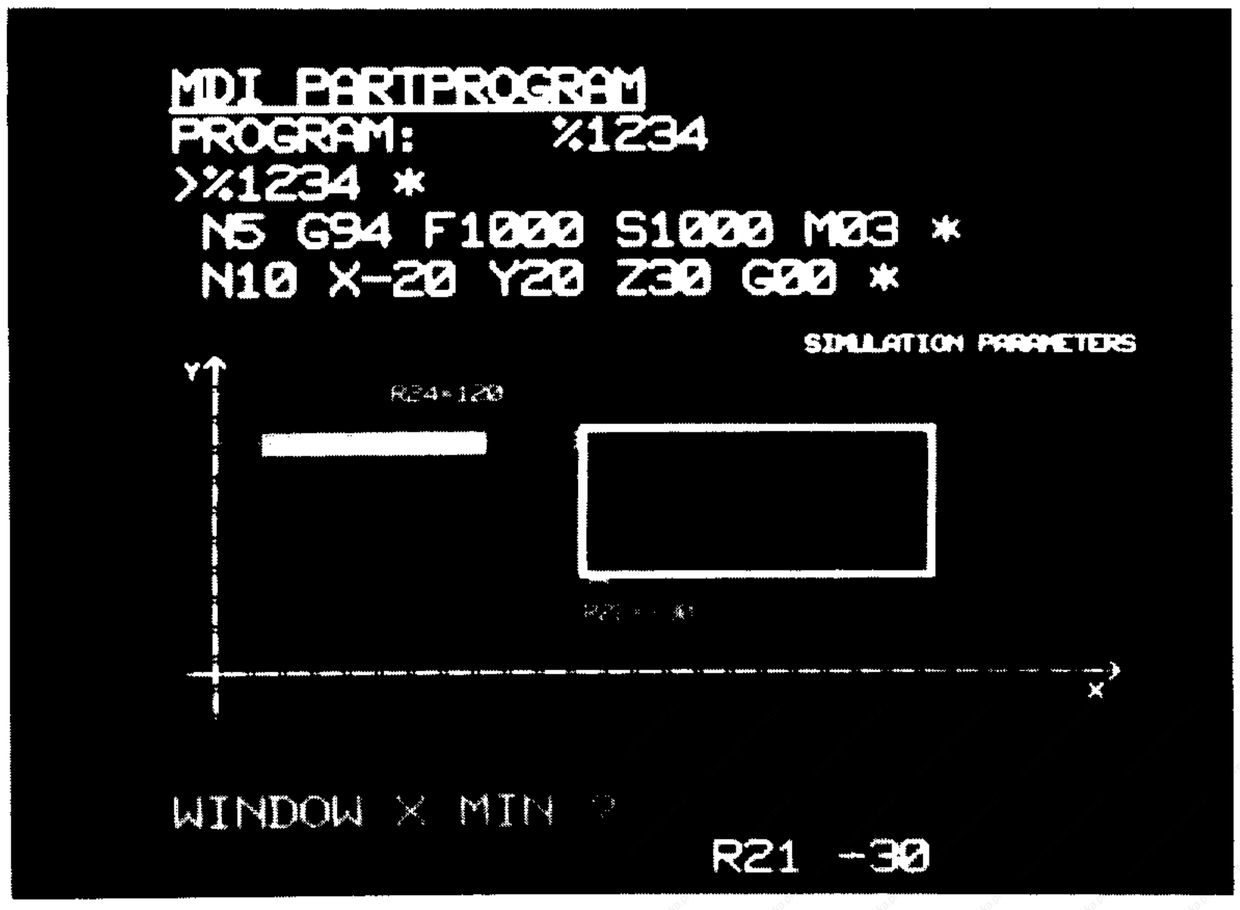
The display for blank definition now appears.



Input the following values as blank dimensions according to the drawing for program %1234

R13	X-MIN	0 mm
R14	X-MAX	100 mm
R15	Y-Min	0 mm
R16	Y-MAX	60 mm
R17	Z-MIN	0 mm
R18	Z-MAX	25 mm

The display for the definition of the simulation area now appears.



For a complete representation of the program example, input following values.

R21	simulation range	X-MIN	-30 mm
R22	simulation range	Y-MIN	-30 mm
R24	simulation range	X-MAX	120 mm
R25	simulation range	Y-MAX	80 mm

For a detail magnification reduce the values for the simulation area accordingly.

The display for the definition of the effective cutter length now appears.

R 27 0

1

Input the number of the tool offset memory for which the cutter length (effective tool length) is to be assigned. In the program example, the machining was done with D01.

R 27 1

with **YES**

Accept input value

R 28 0

3 **0**

Assign the effective cutter length, e.g. 30 mm, to the tool offset memory by pressing and accept input or preset value.

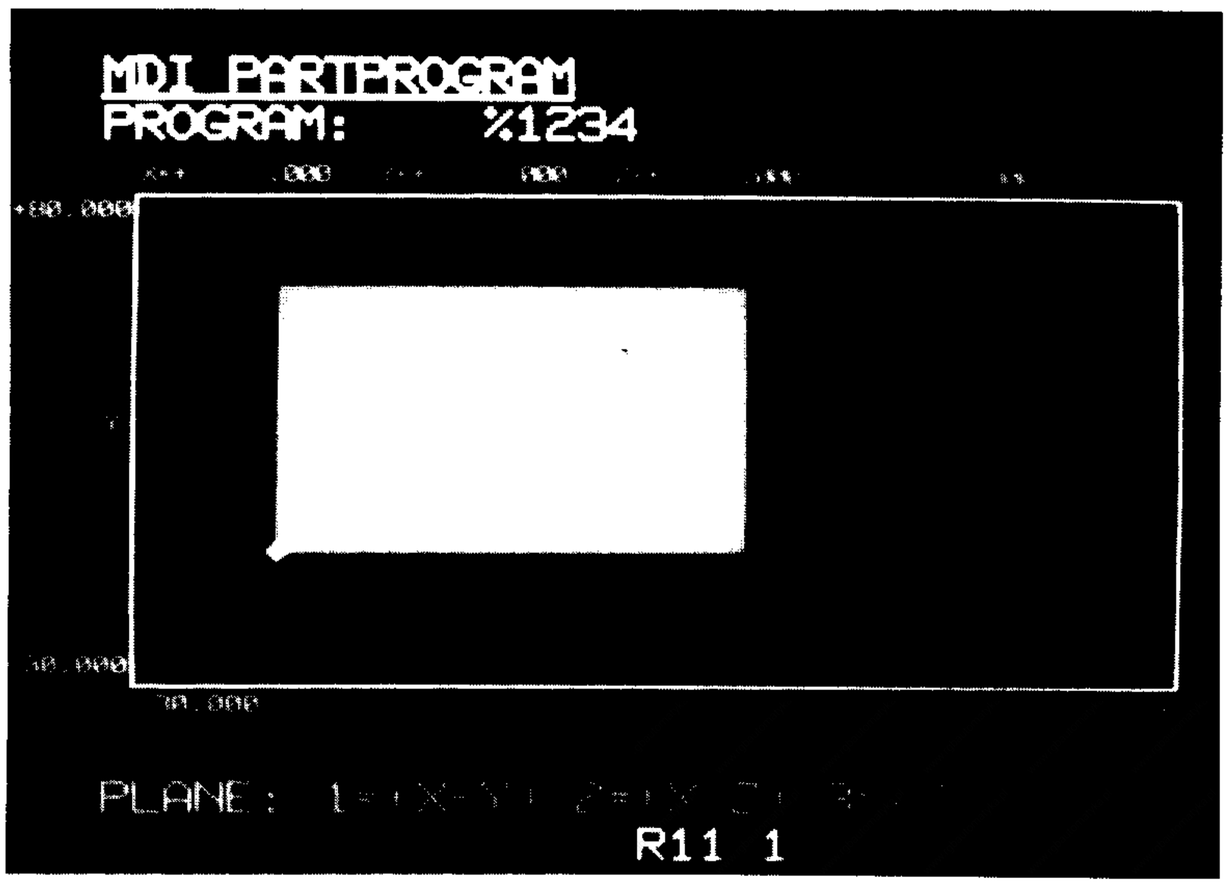
with

Confirm the remaining display contents

with **YES**

Confirm the simulation parameters input

The basic simulation display appears in the X-Y plane.



R 11 1

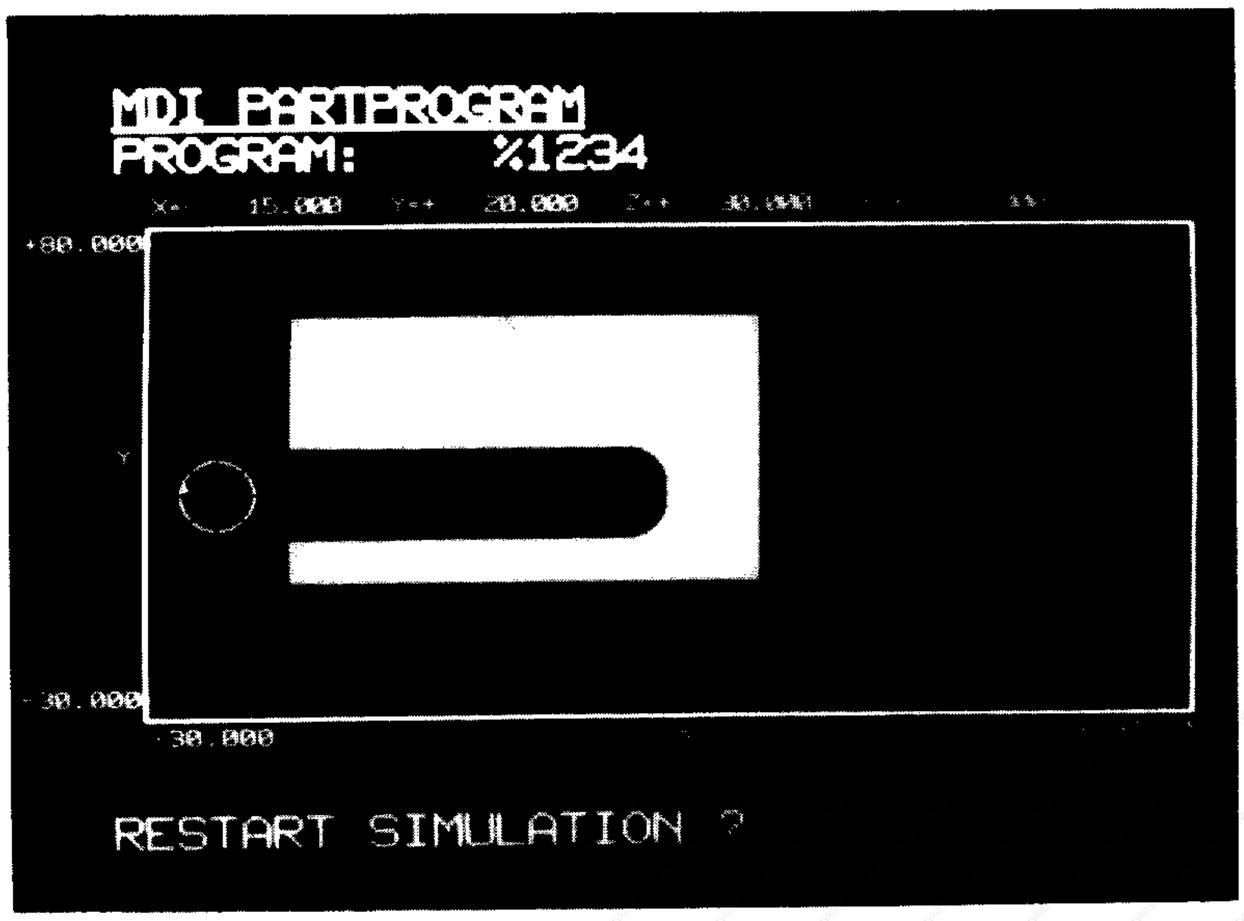
YES

Confirm selected simulation plane or input another simulation plane (2, 3) and confirm with YES

with

Start program simulation

The following display appears:



with

YES

Continue if the other two planes are also to be simulated

R 11 1

2

Select new plane by pressing 2 and confirm with

YES

R 11 2

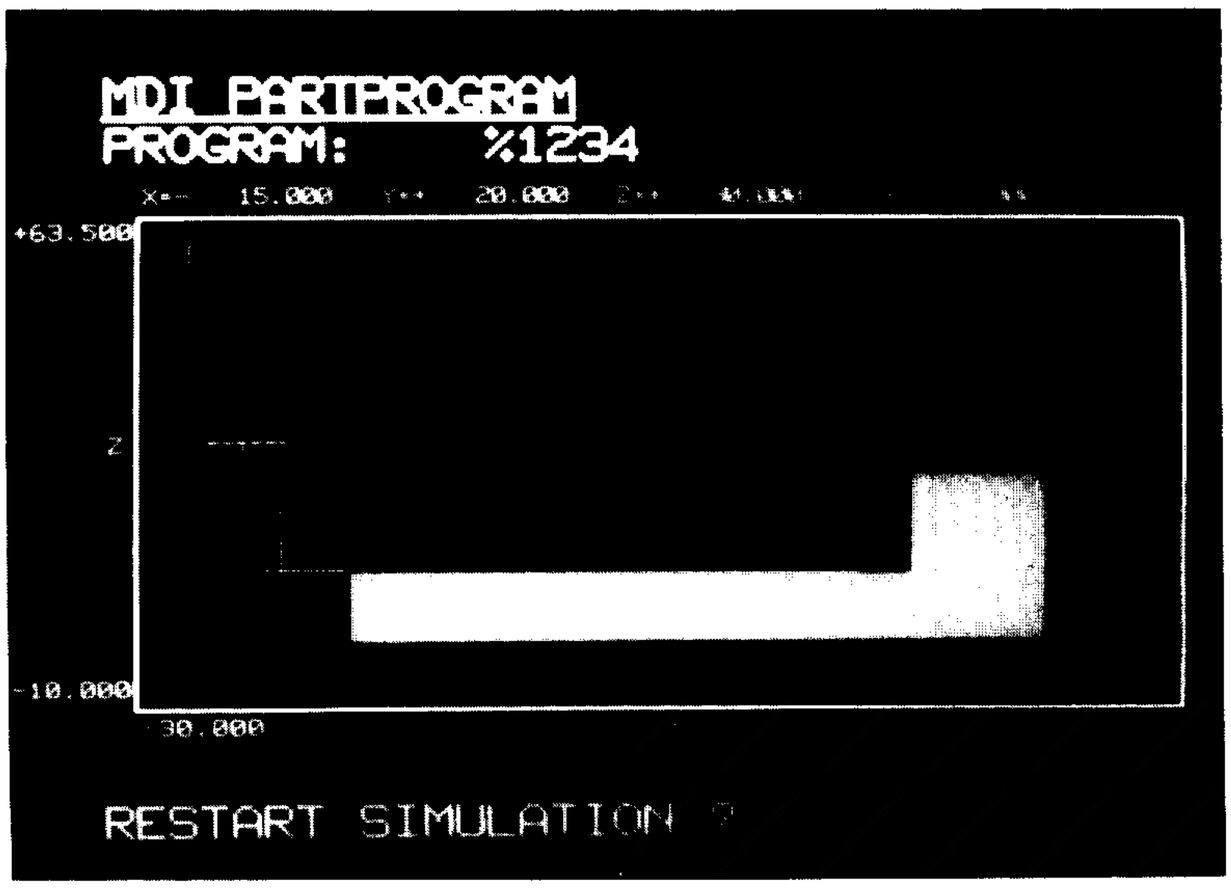
YES

Confirm selected simulation plane

with

Start program simulation

The following display appears.



with

YES

R 11 2

3

R 11 3

YES

with

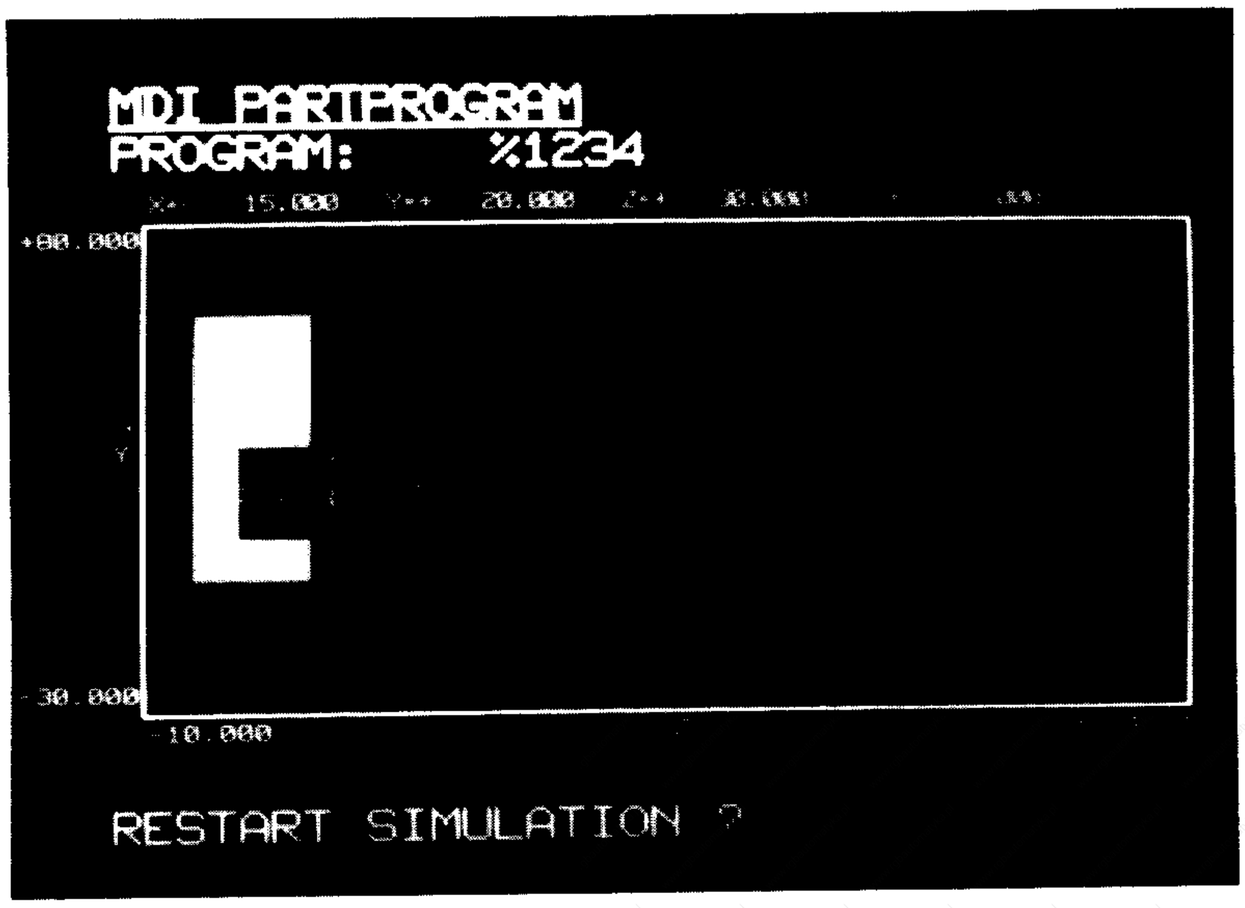
Continue if the third plane is also to be simulated

Select new plane (Z-Y) and confirm with YES

Confirm selected simulation plane

Start program simulation

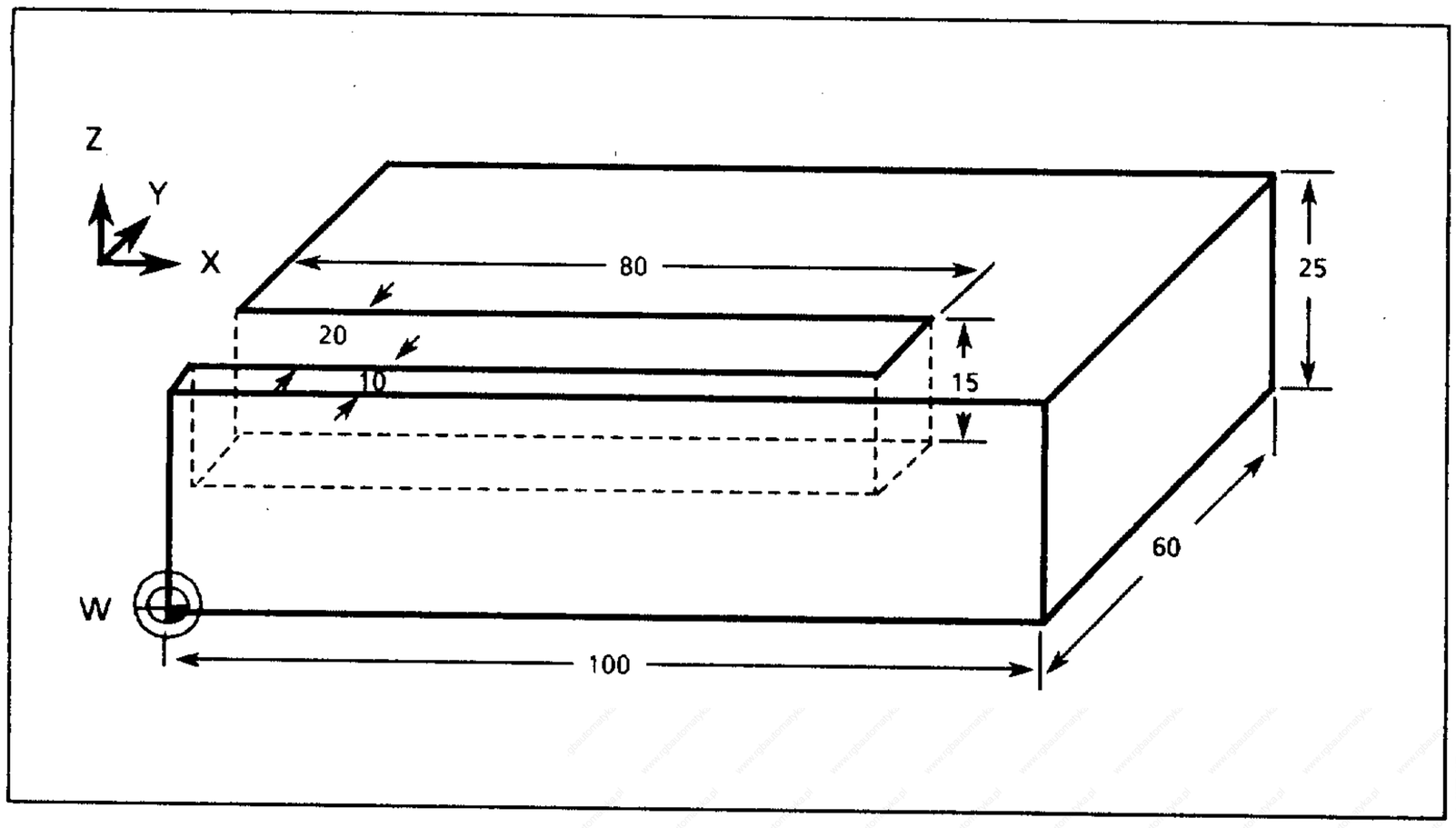
The following display appears.

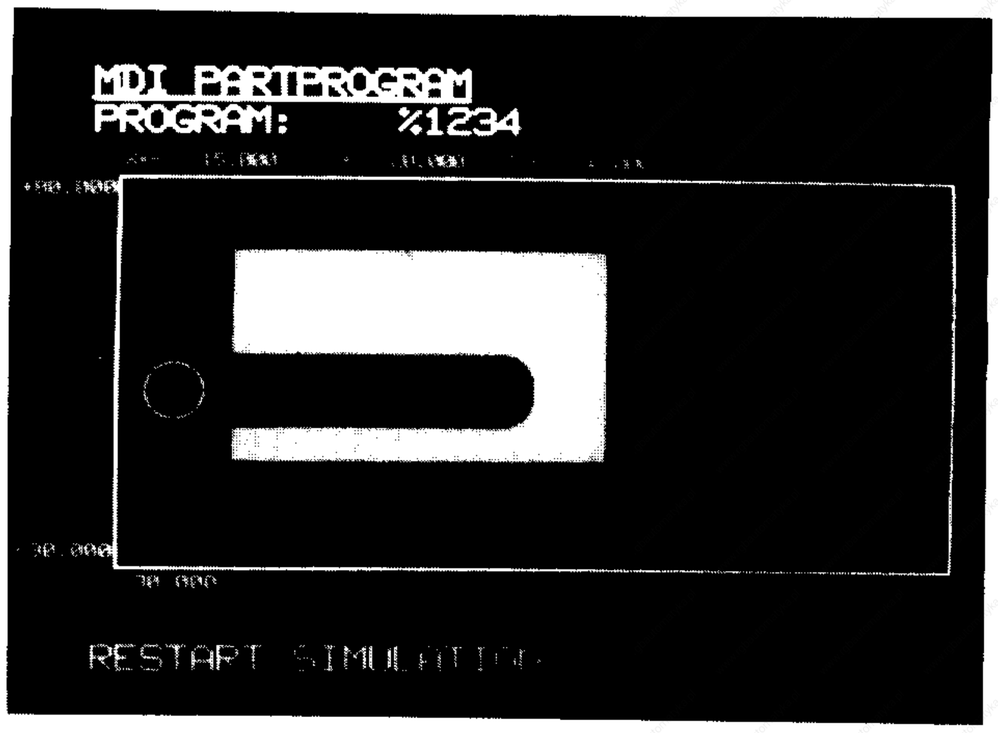


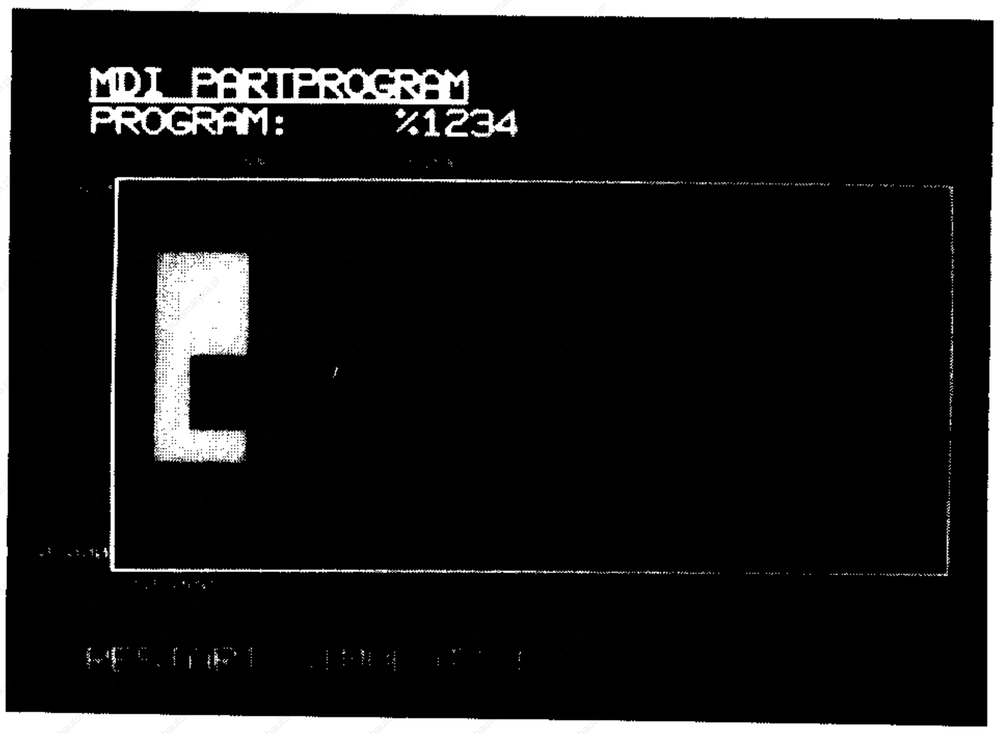
with Return to basic menu

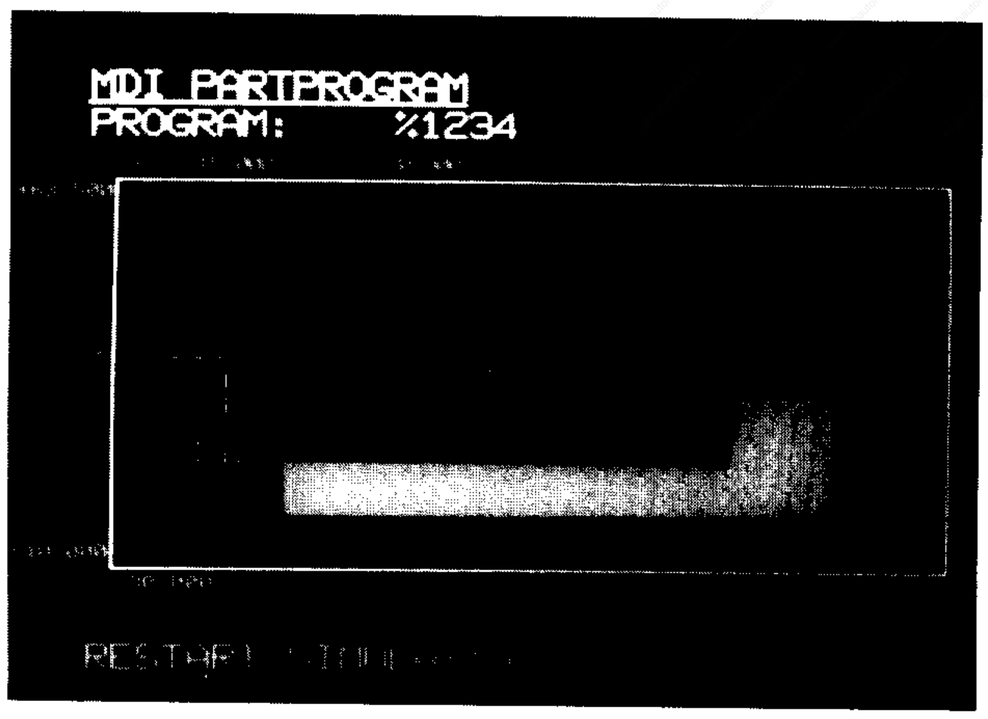
NO

Simulated program example %1234 (3M) in all 3 planes



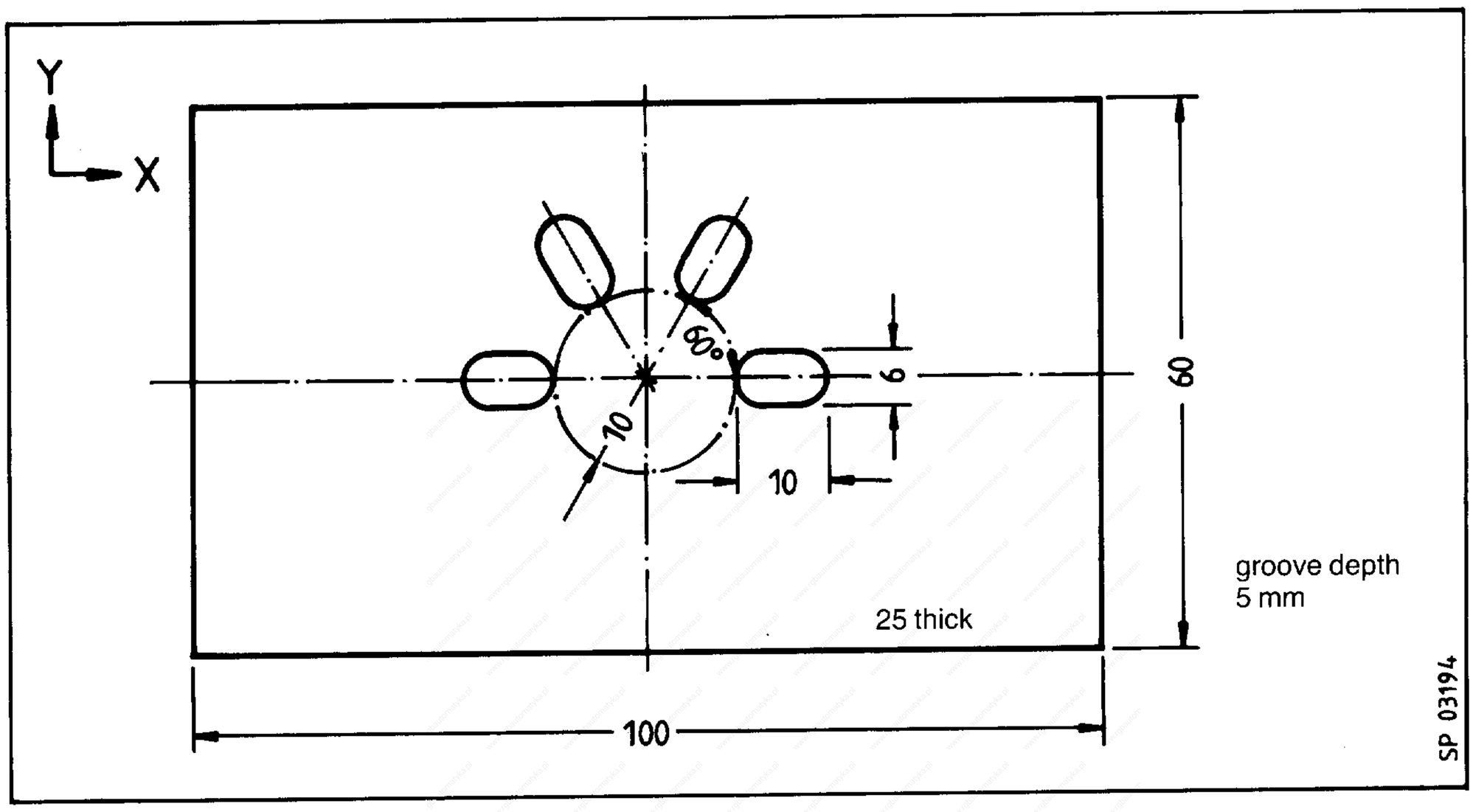






11.3.3 Program Example 2 (3M) (Milling pattern groove)

The following program section shows the parameter assignment of the grooving cycle L94 with graphic support.



Cutter radius 2 mm

Part program created with operator guidance

```
%1235
N5 G94 F1000 S1000 M03
N10 D02 Z30 G00
N15 R22 50 R23 30 R24 8 R25 0
N20 R26 60 R27 4 R12 6 R13 10
N25 R01 5 R02 25 R03 20
N30 L94
N35 M30
```

11.3.3.1 Operation Sequence for Program Example 2 (3M)

Activate operator guidance (see 1.6).

The program is created with operator guidance and is assigned the number %1235.

The program is opened and block N5 is entered as described for example 1 (3M Section 11.3.2.2)

11.3.3.2 Input block N10

N10

YES	Store offered block number
YES	Select "GEOMETRIE/GEOMETRY" menu

with

NO + **YES**	Select "VORBEREITENDE FUNKTIONEN/PREPARATORY FUNCTIONS" menu

with

NO + **YES**	Select "SCHNEIDENRADIUSKOMPENSATION/CUTTER RADIUS COMPENSATION" menu and "SKR-Anwahl/CRC selection" menu

D **0** **2**	Input tool compensation number and store with **YES**

with

NO	return to basic menu

with

YES	Select "GEOMETRIE/GEOMETRY" menu and then linear interpolation via the "INTERPOLATIONSART/TYPE OF INTERPOLATION" menu
YES	Select cartesian coordinates

X

NO

No endpoint X axis

Y

NO

No endpoint Y axis

Z

3 **0**

Input endpoint Z30 and store with **YES**

4.

NO

No endpoint 4th axis

G00

YES

Store offered preparatory function

with

NO

Page through to block end

*

YES

Terminate block

with

NO

Reject "SIMULATION" menu

11.3.3.3 Parameter Input for Grooving Cycle Block N15-N30

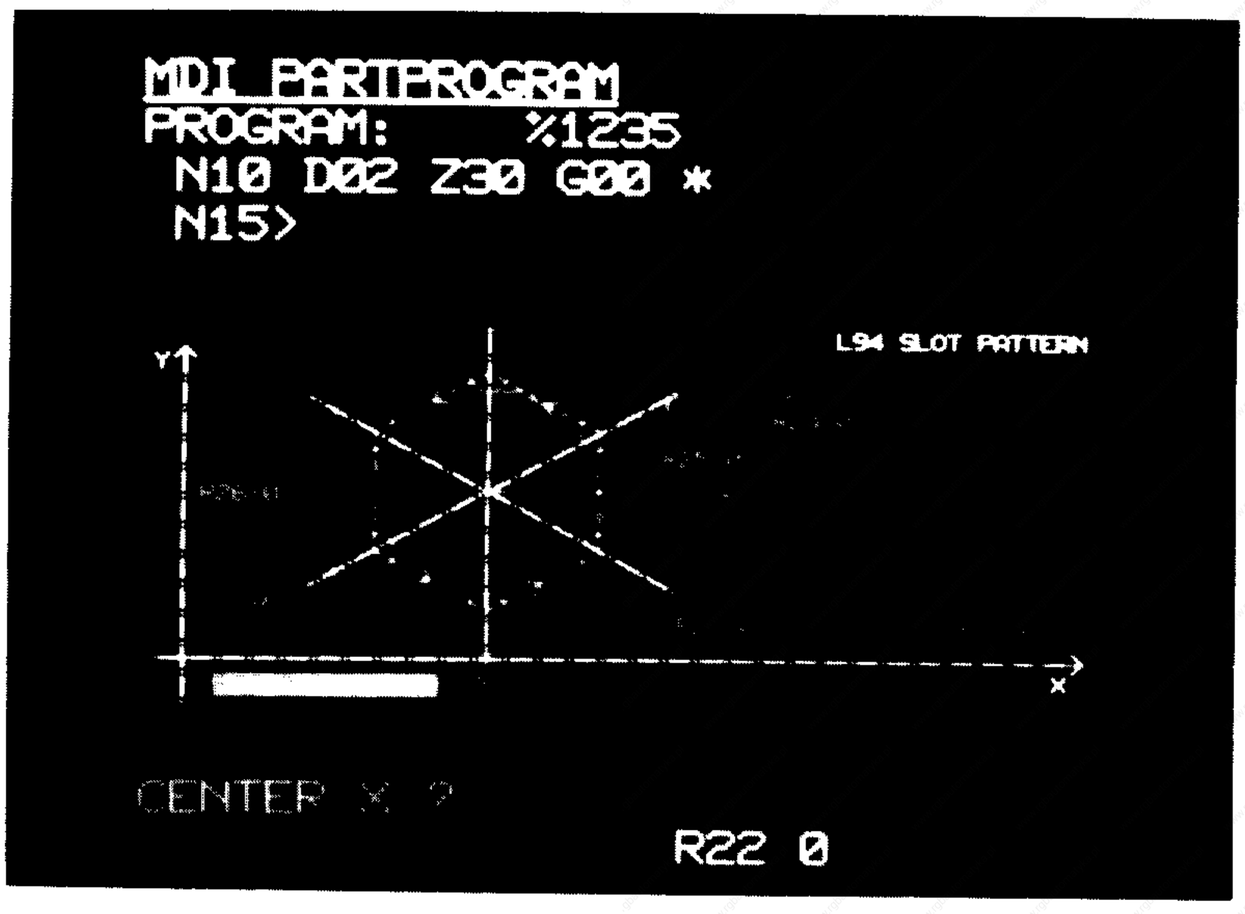
with

NO + **YES** Select "ARBEITSZYKLUS/WORK CYCLE" menu

NO + **YES** Select "FRAESBILDER/MILLING PATTERN" menu

YES Select "L94 FRAESBILD NUT/L94 GROOVING PATTERN" menu

The following display appears.



R 22 0

5 **0**

Input the centre of the milling pattern in X = 50 mm and store with **YES**

R 22 50

R 23 0

3 **0**

Input the centre of the milling pattern in Y = 30 mm and store with **YES**

R 23 30

R 24 0

8

Input radius 8 mm and store with **YES**

R 24 8

R 25 0

YES

Confirm offered starting angle 0 degree

R 26 0

6 | 0

Input indexing angle 60 degrees and store with **YES**

R 26 60

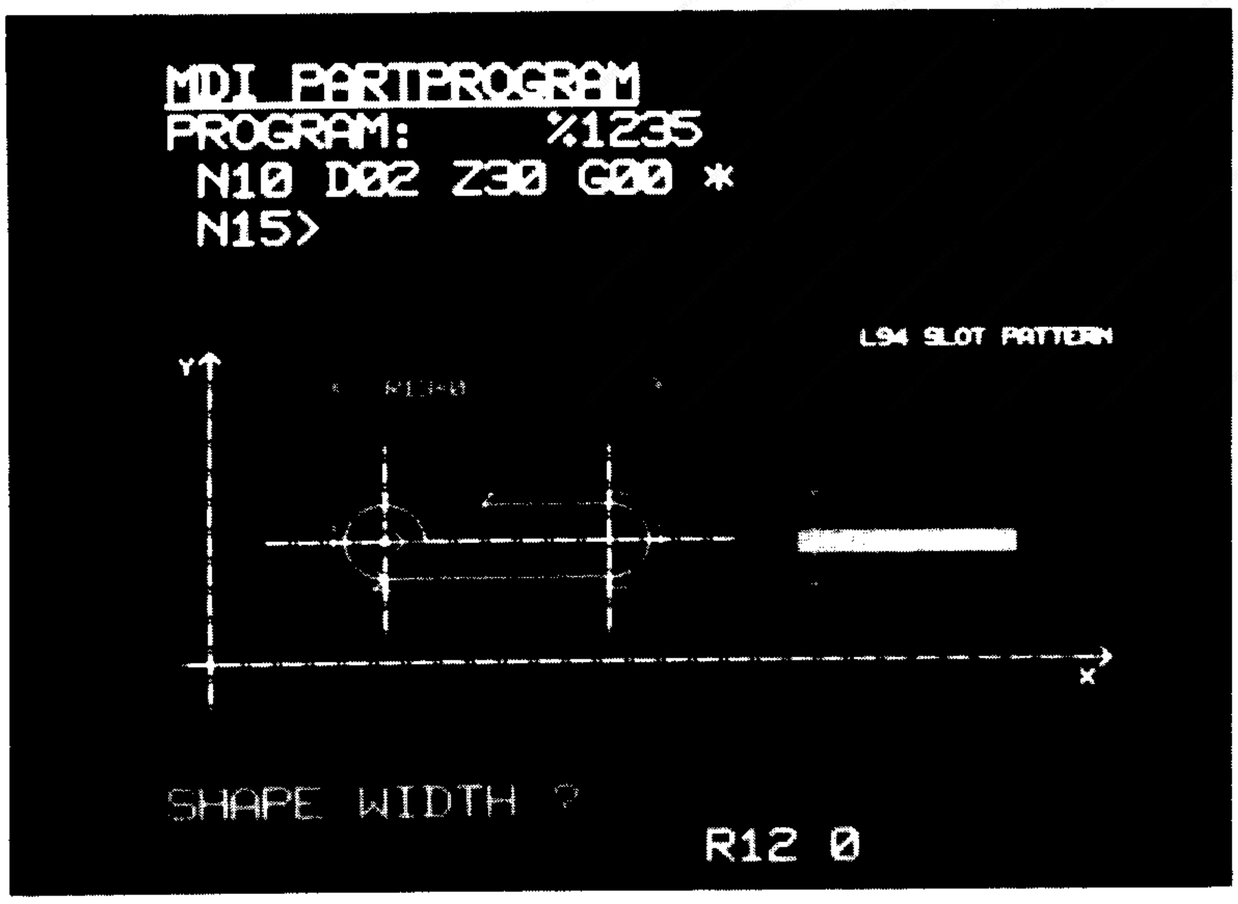
R 27 0

4

Input number of grooves and store with **YES**

R 27 4

The following display appears.



R 12 0

6

Input pocket width 6 mm and store with **YES**

R 12 6

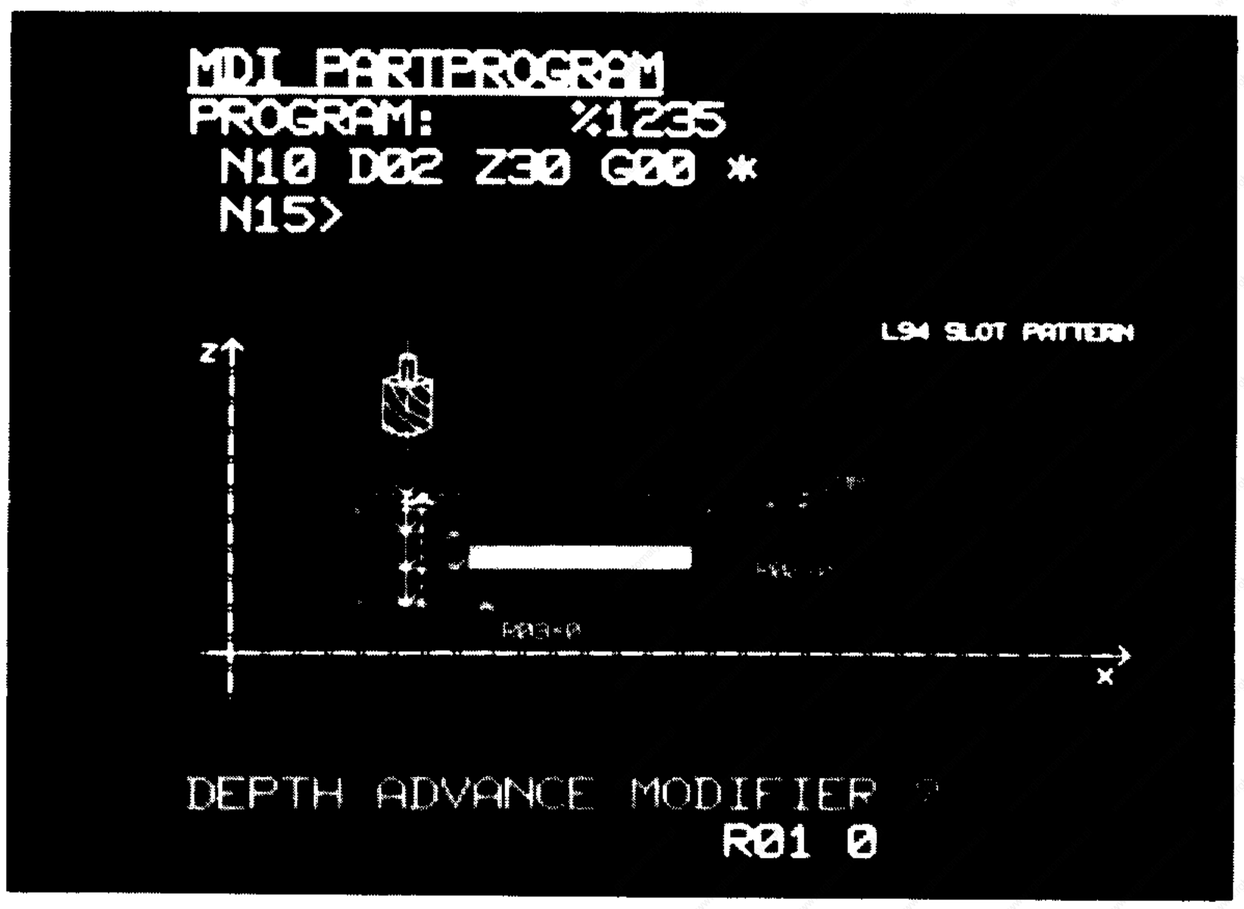
R 13 0

1 0

Input pocket length 10 mm and store with **YES**

R 13 10

The following display then appears.



R 01 0

5

Input degression amount and store with **YES**

R 01 5

R 0 20

2 5

Input reference plane 25 mm and store with **YES**

R 02 25

R 03 0

2 0

Input final drilling depth 20 mm and store with **YES**

R 03 20

with

YES

Continue if all values have been correctly input and confirmed

with

YES

Accept the parameters for the part program, this is done in several blocks from N20 up to N30

N20 to N30

YES

Confirm offered block number and store further parameters

L 94

YES

Store cycle call

*

YES

Terminate block

with

NO

Reject "SIMULATION" menu

11.3.3.4 Input Block N35

N35

YES — Store offered block number

with

NO — Select "SONDERFUNKTION/SPECIAL FUNCTIONS" menu and page through to "PROGRAMMENDE/PROGRAM END" menu

YES — Select "PROGRAMMENDE/PROGRAM END" menu

YES

M30 — Store offered program end

YES

* — Terminate block

YES

with — Continue with the " PROGRAMMSIMULATION/PROGRAM SIMULATION" (see also Section 11.3.3.5).

YES

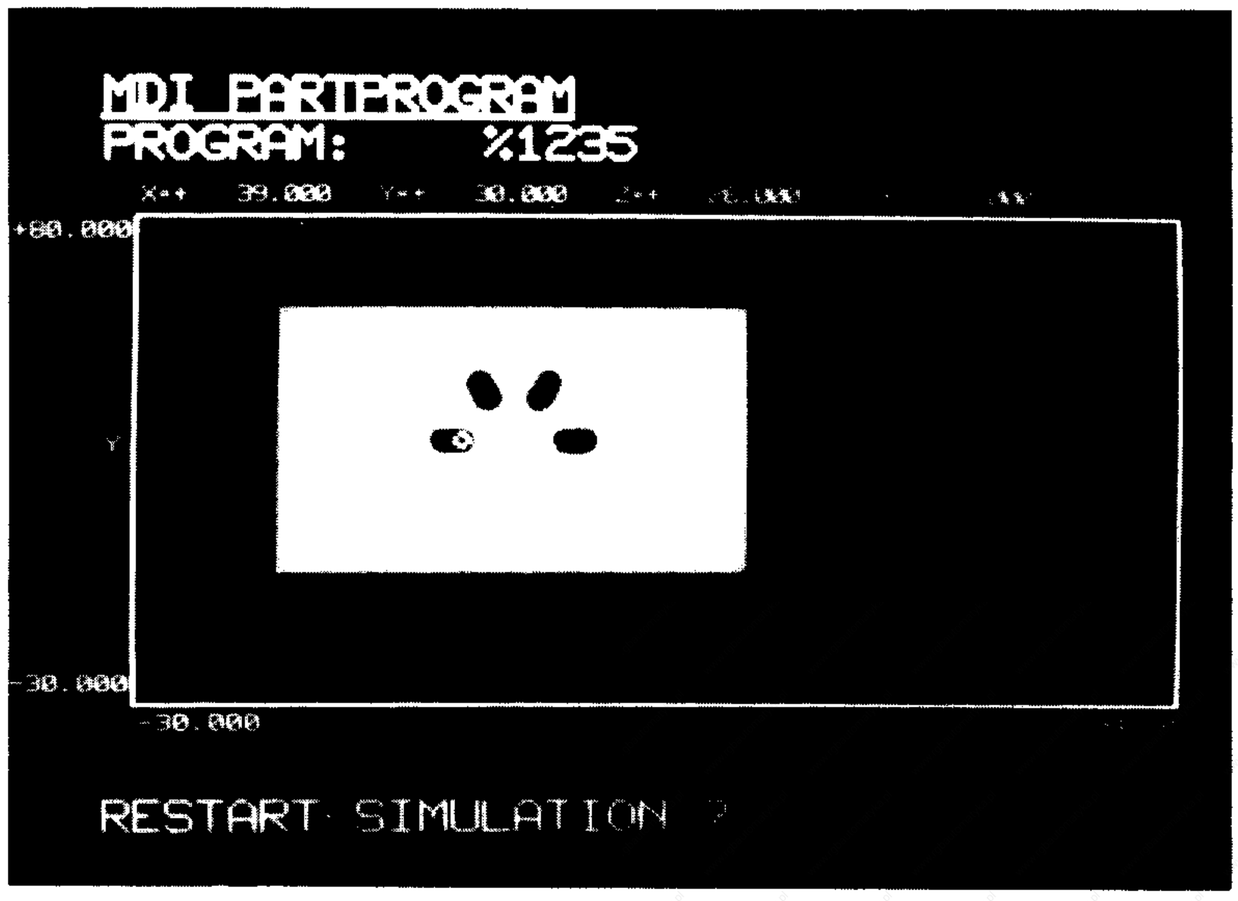
11.3.3.5 Simulation of Program Example 2 (3M)

For simulation of the program example %1235 for the 3M, the simulation parameters must be assigned using operator guidance. In addition, the values for the cutter radius (2 mm) and length must be stored in the tool offset memory DO2.

The simulation parameters must be assigned with the following values using operator guidance (see also Section 11.4.3.2)

R11	Plane selection	1 (X-Y plane)
R33	Type of representation	2 (Optical cutting)
R13	Blank X MIN	0 mm
R14	Blank X MAX	100 mm
R15	Blank Y MIN	0 mm
R16	Blank Y MAX	60 mm
R17	Blank Z MIN	0 mm
R18	Blank Z MAX	25 mm
R21	Simulation area X MIN	-30 mm
R22	Simulation area Y MIN	-30 mm
R24	Simulation X MAX	120 mm
R25	Simulation Y MAX	80 mm
R27	Tool offset (1)	1 (TO memory 1)
R28	Cutter length (1)	20 mm

After confirmation of the simulation parameters input and start of the program simulation, the following display is created.



Simulated program %1235 in X-Y plane

Subsequently it is possible to simulate the program in the other two simulation planes (Z-Y and X-Z). The representation of simulation can be also modified (see also section 11.4.3.2).

11.4 Graphic Simulation (only with Graphics Option)

Graphic simulation is used to test part programs (main programs and subroutines).
It is possible to test not only the syntax but also program execution. Programs may not be terminated with M02, M30, or M17. In this way, simulation may take place during programming after every block. Part programs to be simulated must be stored in the part program memory of the control.

Graphic simulation can only be used in MDI-PP mode under control of operator guidance. Simultaneous operations such as simulation and automatic mode or programming and automatic mode are not possible.

Graphic simulation is always traversing in relation to the workpiece. The first block of a program is not displayed since in this block the control performs the conversions from machine reference system to workpiece reference system.

Any existing zero offsets are eliminated and the tool length offsets are calculated so that the path of the cutter in relation to workpiece zero can be displayed.

The tool path is always displayed in relation to workpiece zero. For the 3T turning machine, the path of the cutting tool tip is displayed. For the 3M milling machine control, the path of the cutter centre is represented (line graphics). In addition, in the case of 3M, it is possible to select graphic simulation as "optical cutting". The tool geometry is then represented accordingly and taken into consideration during simulation.

Graphics simulation requires a feedrate for decoding and program display. Therefore, it is recommended to first define the technology values for a newly opened program (such as feedrate and spindle speeds) in order that the axis movements then entered can be simulated. In all other cases, an alarm appears if the simulation is started (F word not programmed). In subroutines, an F value need not be programmed.

Collisions tests are not carried out in the machine area.

3T

Due to the rotational symmetry of the parts in relation to the Z axis, the semi-symmetrical representation for graphic simulation was selected for the turning machine control. This permits more detailed information to be shown.
Accordingly, all movements are represented in the positive area behind the turning centre.

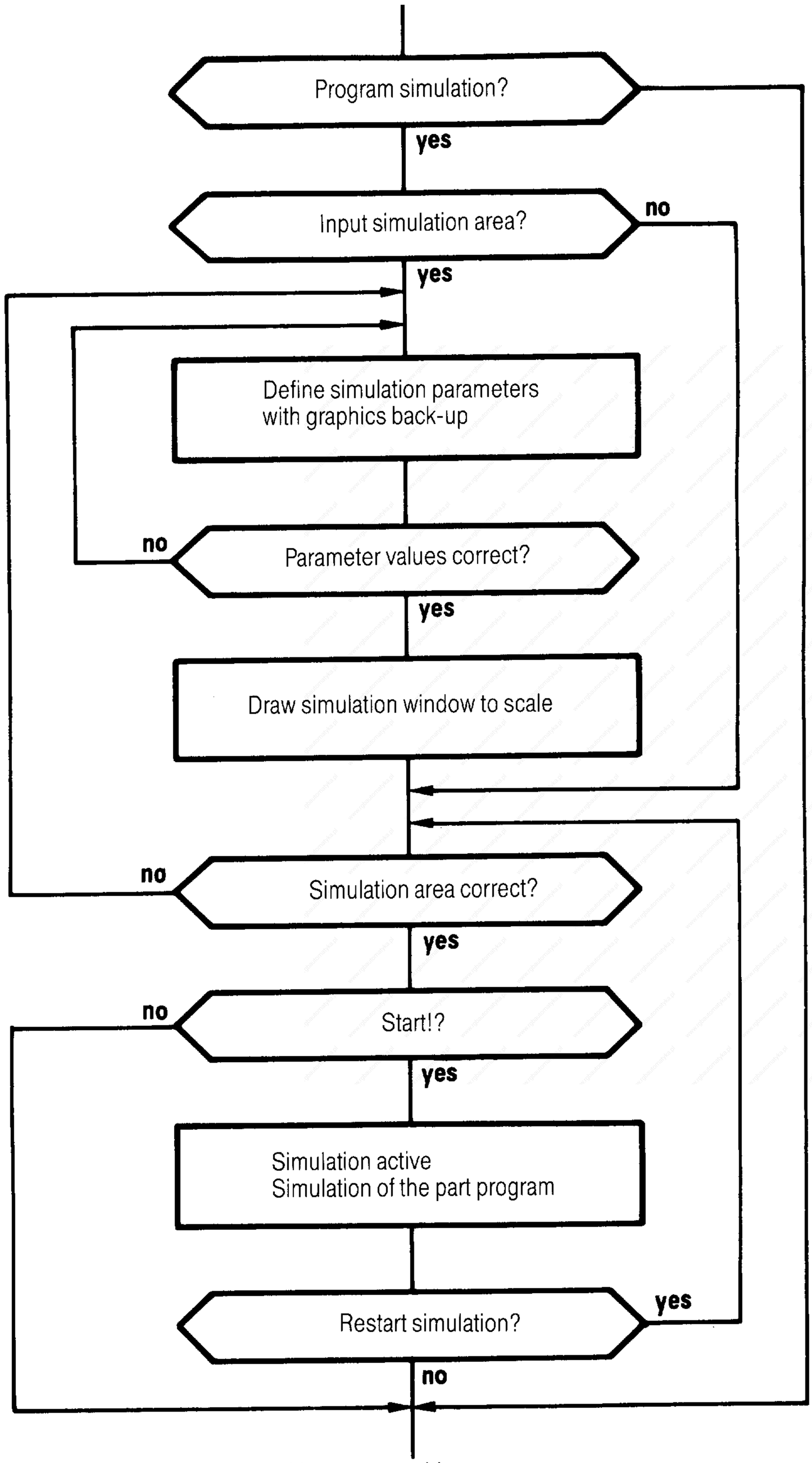
3M

For the milling machine control, single plane representation was chosen as graphic simulation. Each of the three planes may be simulated individually. Three-dimensional representation is not possible.

11.4.1 Sequence of Graphic Simulation

11.4.1.1 Sequence for 3T

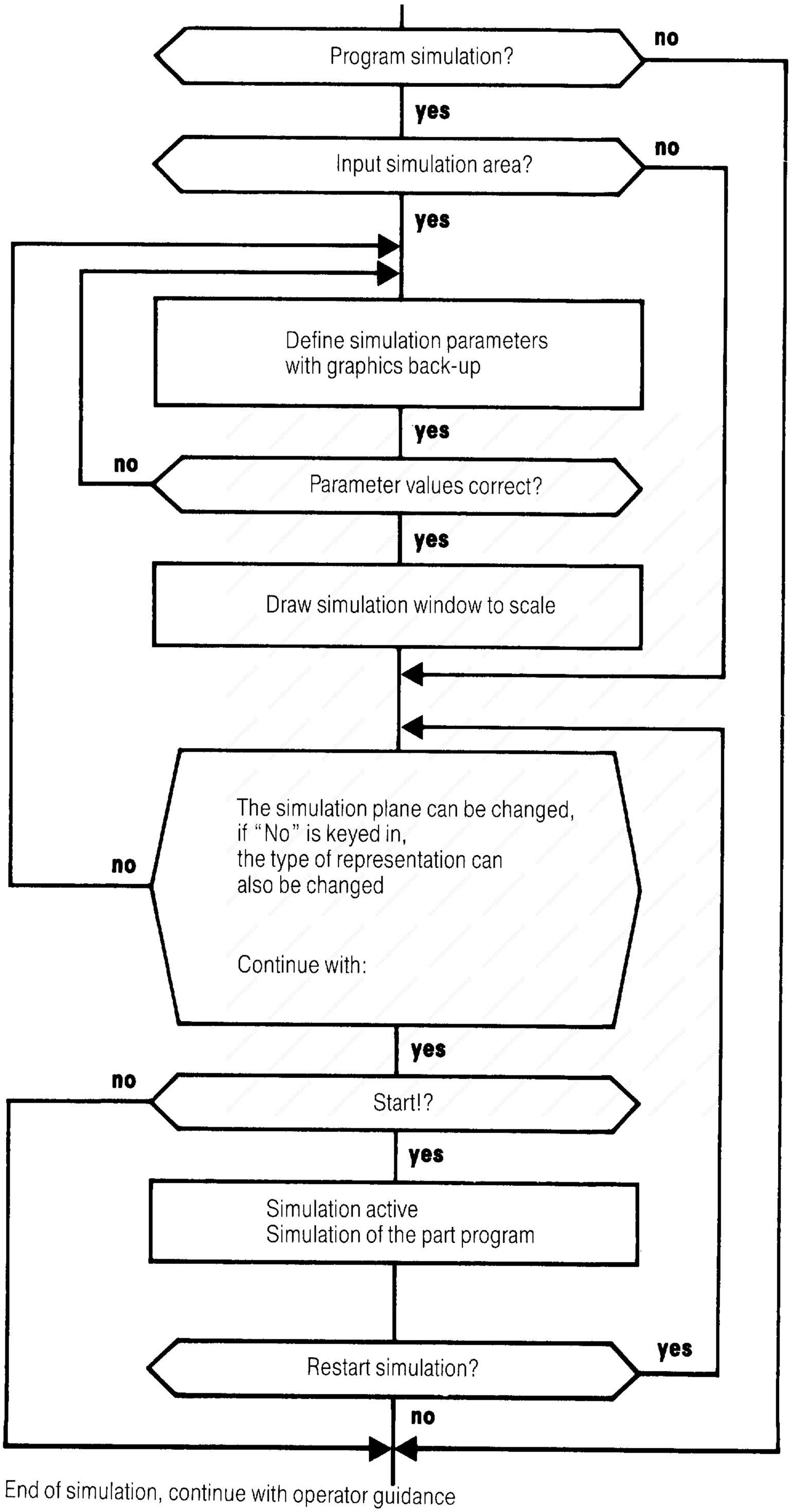
Simulation is offered in operator guidance



End of simulation, continue with operator guidance

11.4.1.2 Sequence for 3M

Simulation is offered in operator guidance



SP03235.0

11.4.2 Influence of Simulation of the Interface

The output of the machine signals M, S, T, H, rapid traverse, thread, etc. can be activated during graphic simulation by setting a machine datum. In this case, the output is made without consideration of any existing machine functions such as the shifting of tool changers, gears, etc.

Since the "EINLESEFREIGABE/READ-IN ENABLE" signal cannot be interpreted by the NC during simulation, PLC interlocks cannot be activated in the NC.

Therefore, simulation cannot be used for testing machine functions.
The "Output of machine signals M, S, T, H" function is meant to display the output of these signals via pilot lamps in the case of training installations.

11.4.2.1 Output Signals of the NC

The following signals are not influenced by simulation, irrespective of whether the simulation takes place with or without the output of auxiliary functions.

Spindle stop
Spindle in command area
Spindle speed limitation
Direction of spindle rotation CW
NC-Ready 1
NC-Ready 2
NC Alarm
Arrival at reference point (1st - 4th axis)
M02/M30 Reset

At the end of simulation, "RESET" is generated and output.

11.4.2.2 Output Signals of the NC during Simulation without Auxiliary Function Output

There is no output of additional signals at the NC-PC interface such as:

Traverse commands
Programmed hold (MO)
Thread
Rapid traverse
Program running
G96 selected
Spindle position reached
Auxiliary functions (M, S, T, H) with modification signals

11.4.2.3 Output Signals of the NC during Simulation with Output of Auxiliary Functions

During simulation, the following signals are additionally output (according to program) at the NC/PLC interface:

Programmed stop MOO
Thread
Rapid traverse
Program running
G96 selected
Auxiliary functions (M, S, T, H) with modification signals

No traversing commands are output.

11.4.2.4 Input Signals of the NC

During simulation, the following NC input signals are effective irrespective of whether the simulation takes place with or without the output of auxiliary functions.

Machine control panel

- Skip block
- Single block

Interface signals

- * Emergency stop
- Reset
- Mirroring Z axis (for 3T)
- Mirroring X, Y, Z axis (for 3M)

All other signals are not evaluated by the NC.

S i n g l e b l o c k

is effective during simulation automatic mode. The block change, however, is made by using the "Start" key.

11.4.3 Defaulting Simulation Parameters

The simulation parameters define the simulation area and the blank. Simulation parameters are entered by means of operator guidance with graphics support in the simulation basic display.

The location of the simulation area (window) can be changed according to the dimensions of the part or detail to be represented by using the simulation basic display. This also provides the possibility of setting scaling factors for the window (magnification, reduction, extracts). In addition, it is possible to define a blank. The blank is displayed during graphic simulation.

The simulation parameters are entered with R parameters in the same way as cycle parameters which, however, have only symbolic meaning and are only used as aids. The values input are stored under the corresponding R parameter numbers.

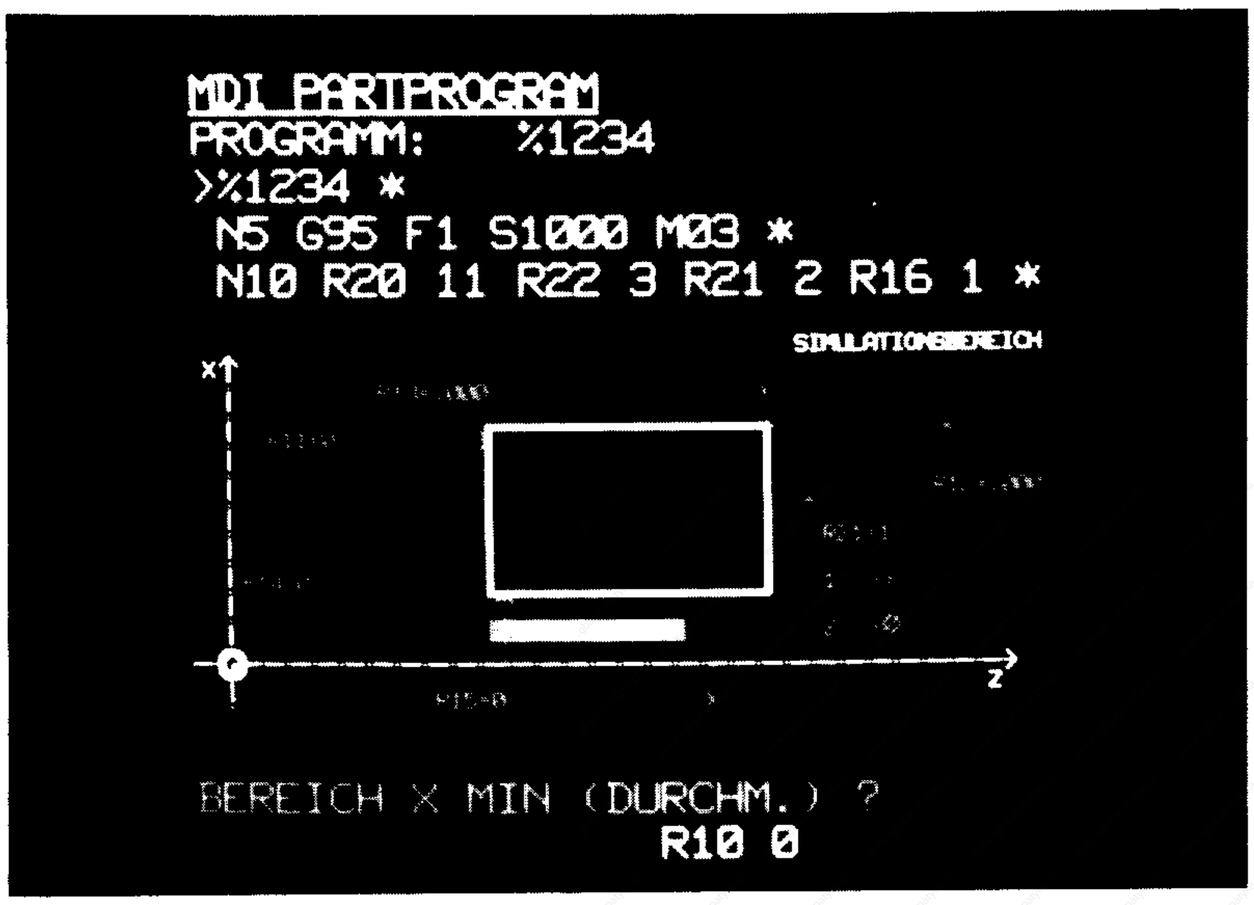
To store all parameters from one display press

The parameters must be defined after switching on. They remain active even after the control is switched off unless modified.

During the input, the respective actual value is displayed; it can be directly accepted by pressing "YES", it may be overwritten without previous deletion and then stored with "YES".

11.4.3.1 Simulation Parameters (3T)

For turning machine controls, there is a basic simulation display for parameter assignment.



Parameter meaning:

R parameter	Texts for guidance	Input format	Preset
R10	Range X min (diameter)	±5.3	Old value
R11	Range Z min (+ or -)	"	Old value
R12	Range X max (diameter)	"	Old value
R13	Range Z max (+ or -)	"	Old value
R14	Workpiece diameter or radius value	"	0
R15	Workpiece length (+ or -)	"	0
R16	Workpiece input	-1.0 = 1 Radius input = 2 Diameter input	1

Dimensioning of the Simulation Area: R10 to R13

For the simulation area, the left lower point and the right top point of the respective plane are defined. All dimensions are related to the workpiece zero.

Due to the rotational symmetry of a turning machine, the X coordinates must always be positive. Negative values are not permitted.
In addition, X values are always input as diameters.

The values for the Z area may be positive or negative depending on whether the workpiece zero is situated on the left side, on the right side or in the centre of the workpiece.

The value range for the simulation area corresponds to the possible traversing path of the control: X = +99 m, Z = ±99 m.

The control optimizes the determined simulation area in such a way that the largest value in the X or Z direction can be represented on the whole display. The smaller value will be adapted accordingly. The adaptation is based on a side relation of X:Z of approx. 1:2.

Workpiece dimensioning: R21, R14, R15

The parameter R21 selects whether the blank is dimensioned in the X direction as a diameter or radius. According to this parameter, the control interprets parameter R14 for the X dimension.

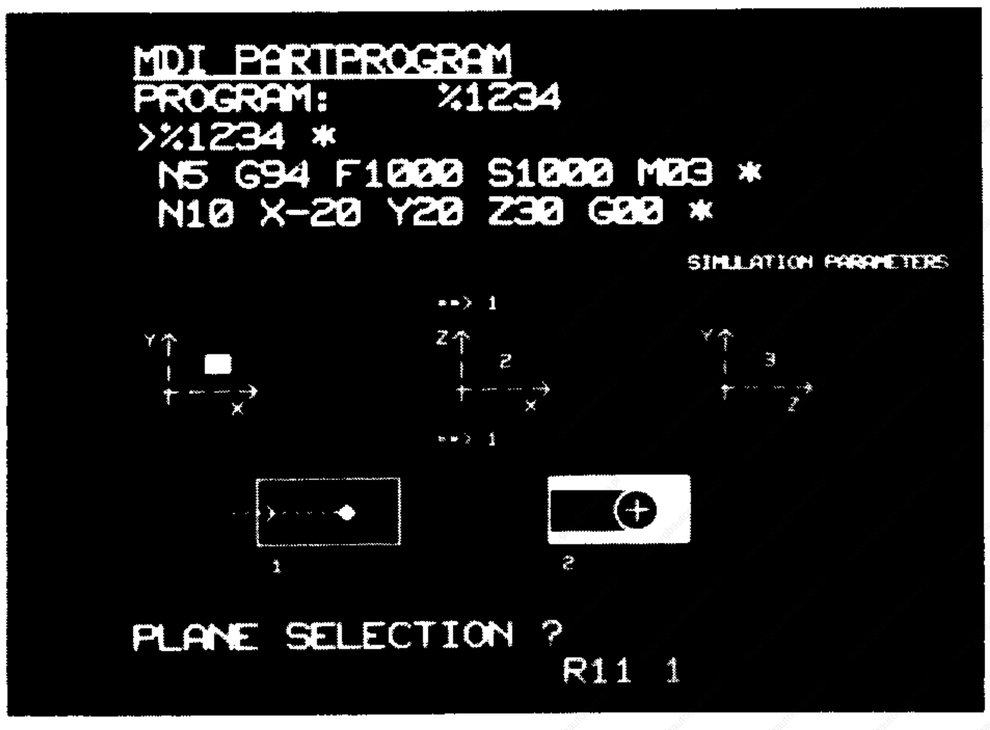
In the Z direction, the length of the blank is determined with R15. This may be entered as a positive or negative value depending on whether the tool zero is left of right of the blank.

11.4.3.2 Simulation Parameters (3M)

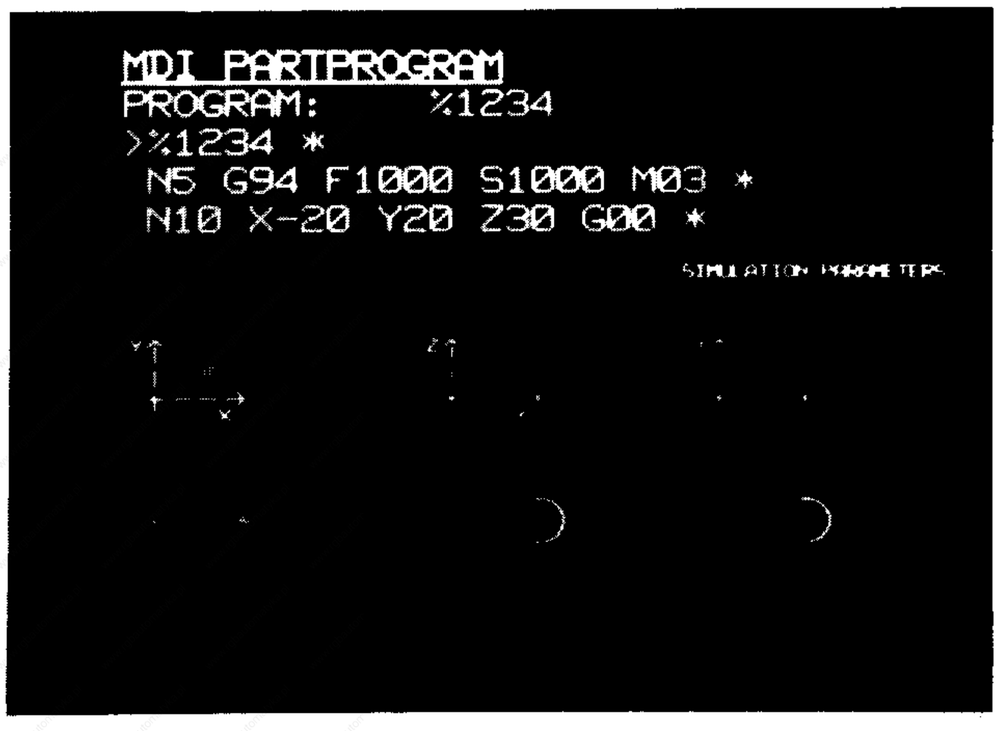
For milling machine controls, the simulation parameters are input in several steps.

Selection of plane and type of representation

for monochrome graphics



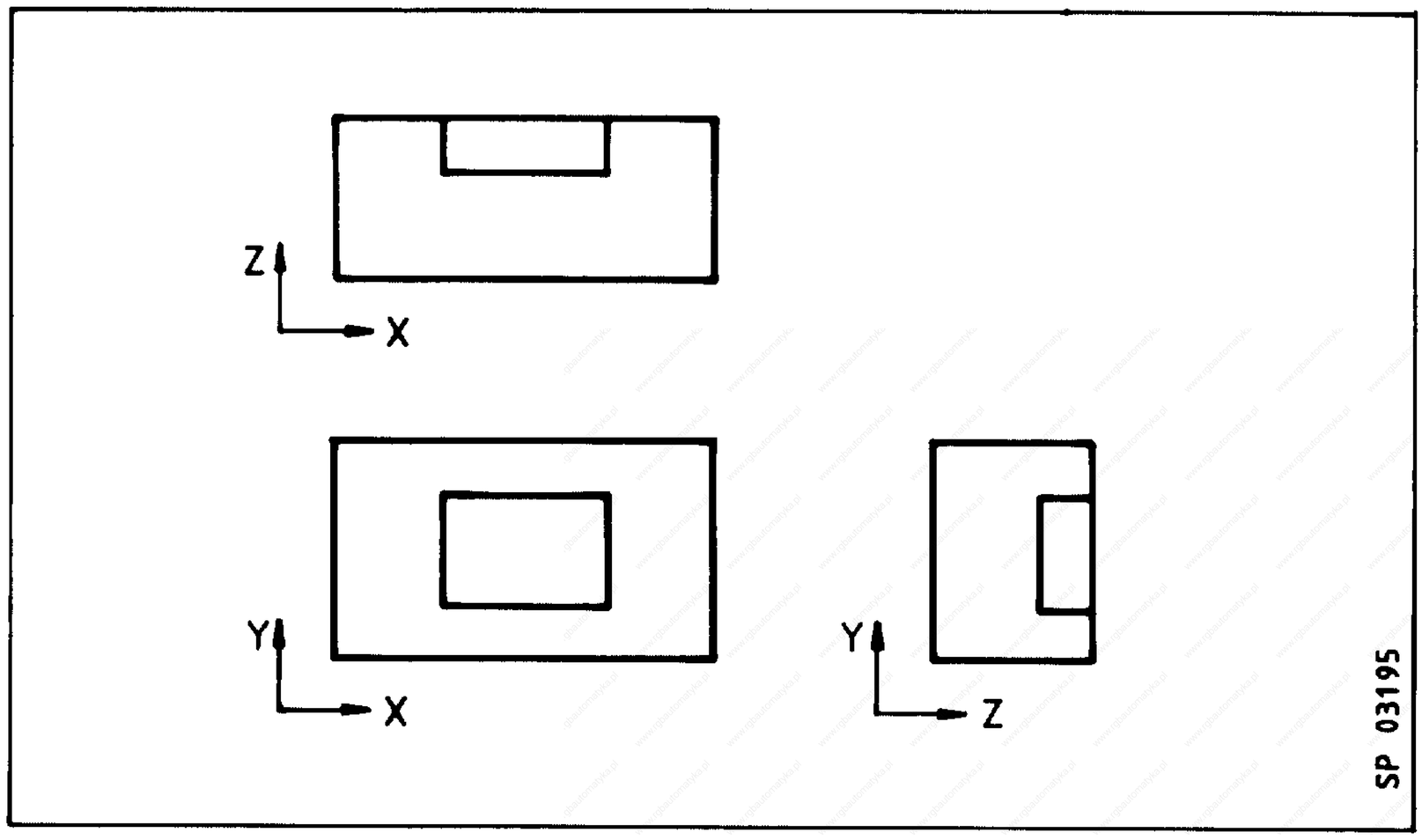
for colour graphics



R parameter	Text for operator guidance	Input format	Preset
R11	Plane selection	+1.0 = 1 = X-Y plane = 2 = X-Z plane = 3 = Y-Z plane	1
R33	Type of representation	+1.0 = 1 = Erase mode = 2 = Erase mode + tool paths = 3 = tool path	1

Plane Selection: R11

R11 selects the viewing plane in which all movements are displayed. The plane arrangement and representation is based on the standard for workshop drawings using the X-Y plane as main view. Accordingly, the following arrangement in the coordinate system results:

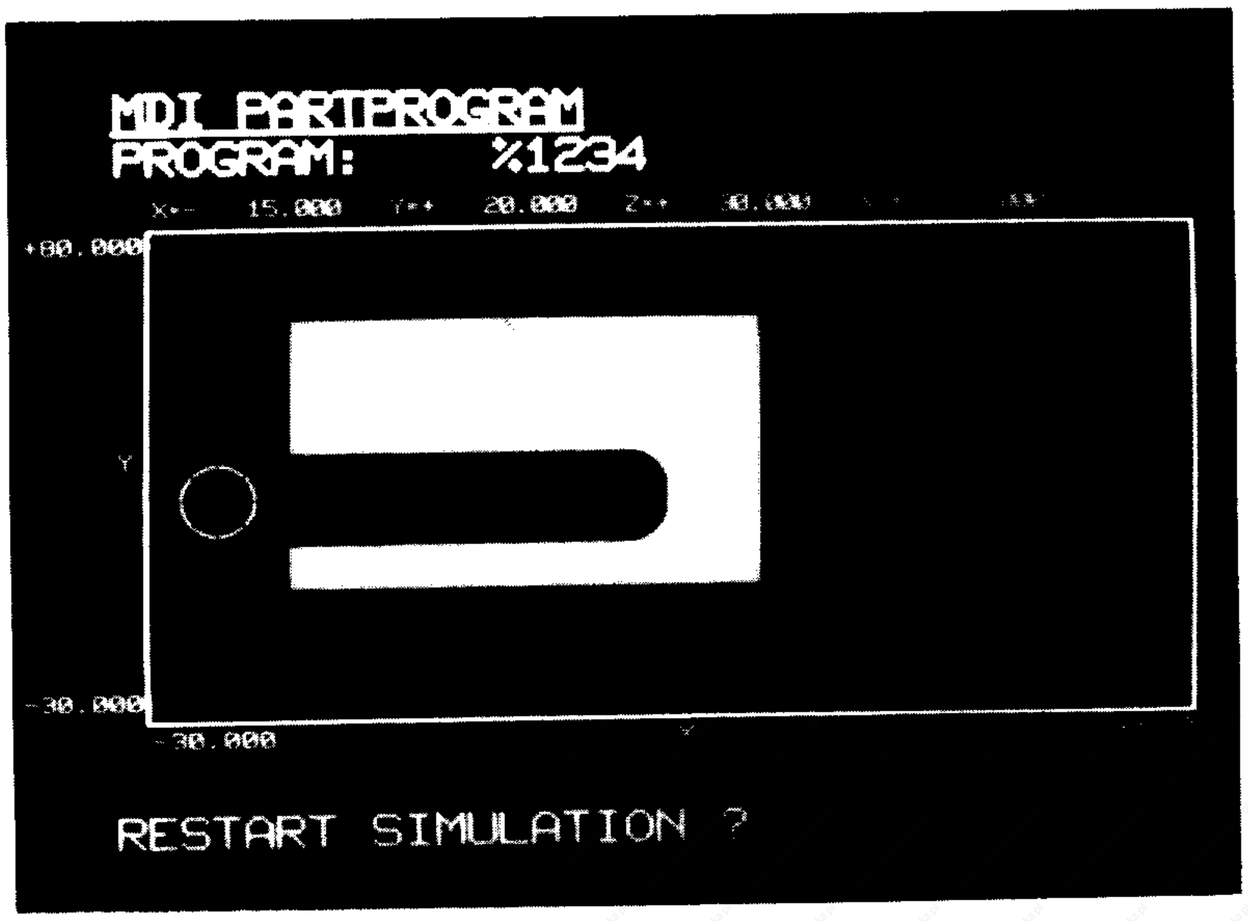


Representation of the Simulation: R33

There are three possibilities of simulation representation:

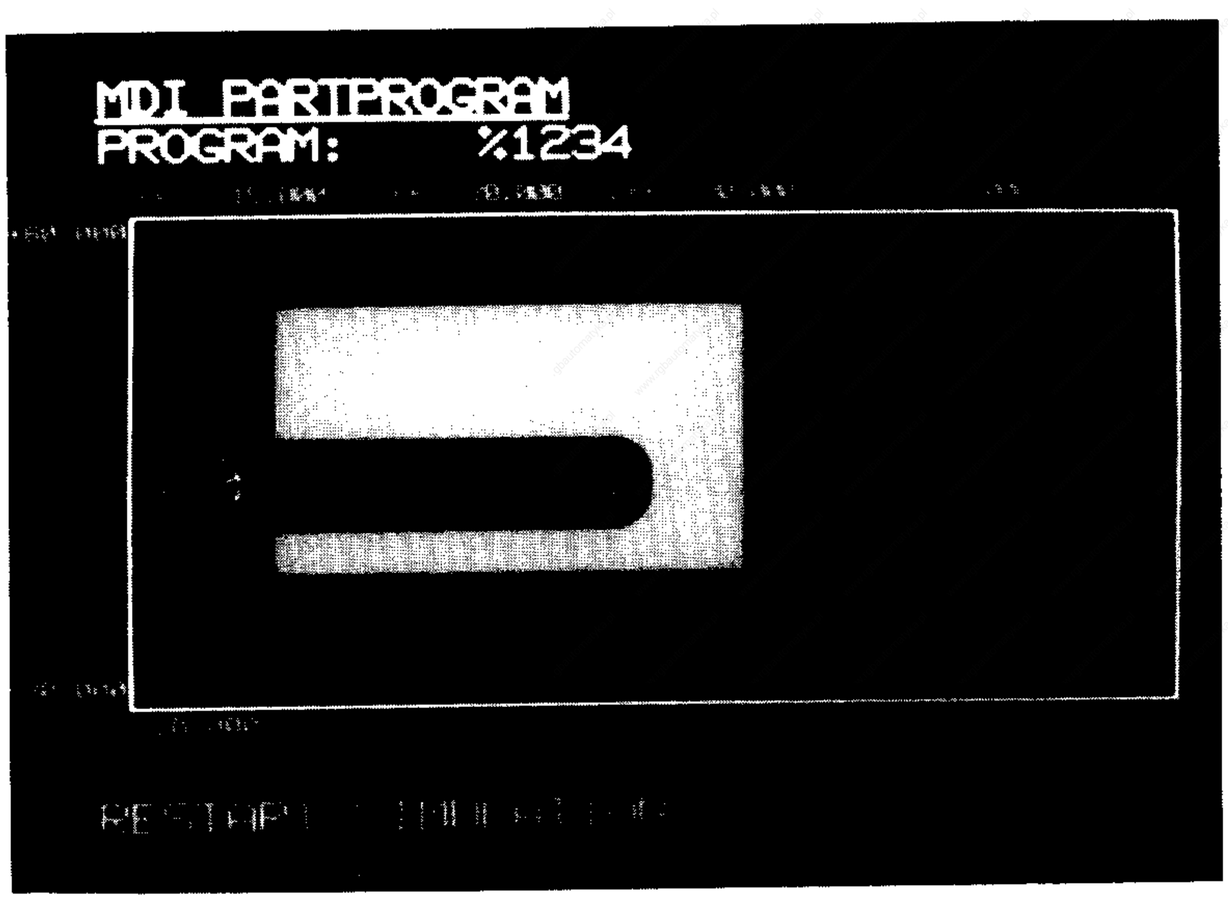
- Optical cutting of the workpiece
- Optical cutting of the workpiece and drawing of the cutter centre path (for colour graphics)
- Representation of the cutter centre path ("line graphics")

Optical Cutting of the Workpiece



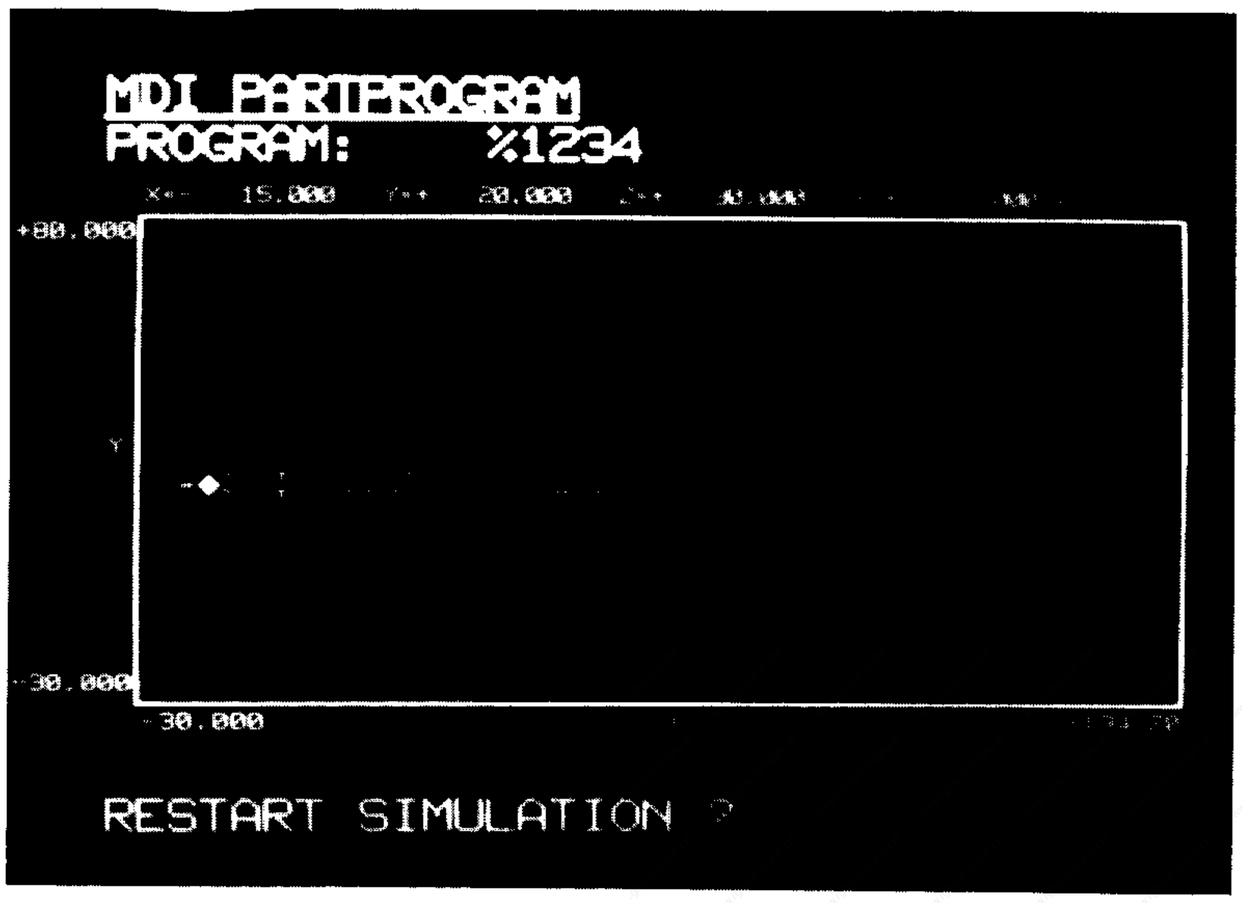
Optical Cutting using the Tool Centre Path

(only colour graphics)

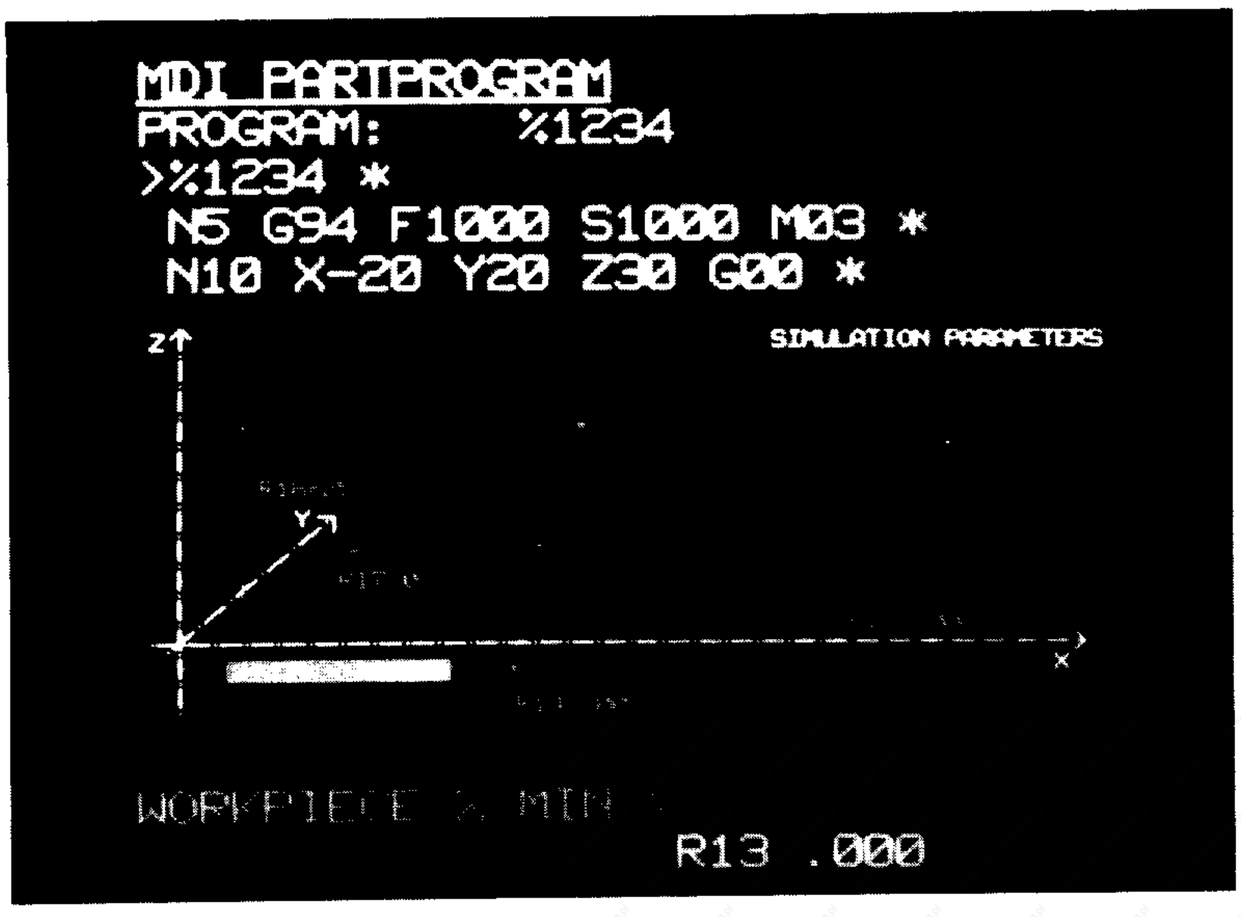


Representation of Cutter Centre Path

("line graphics")



Definition of Blank

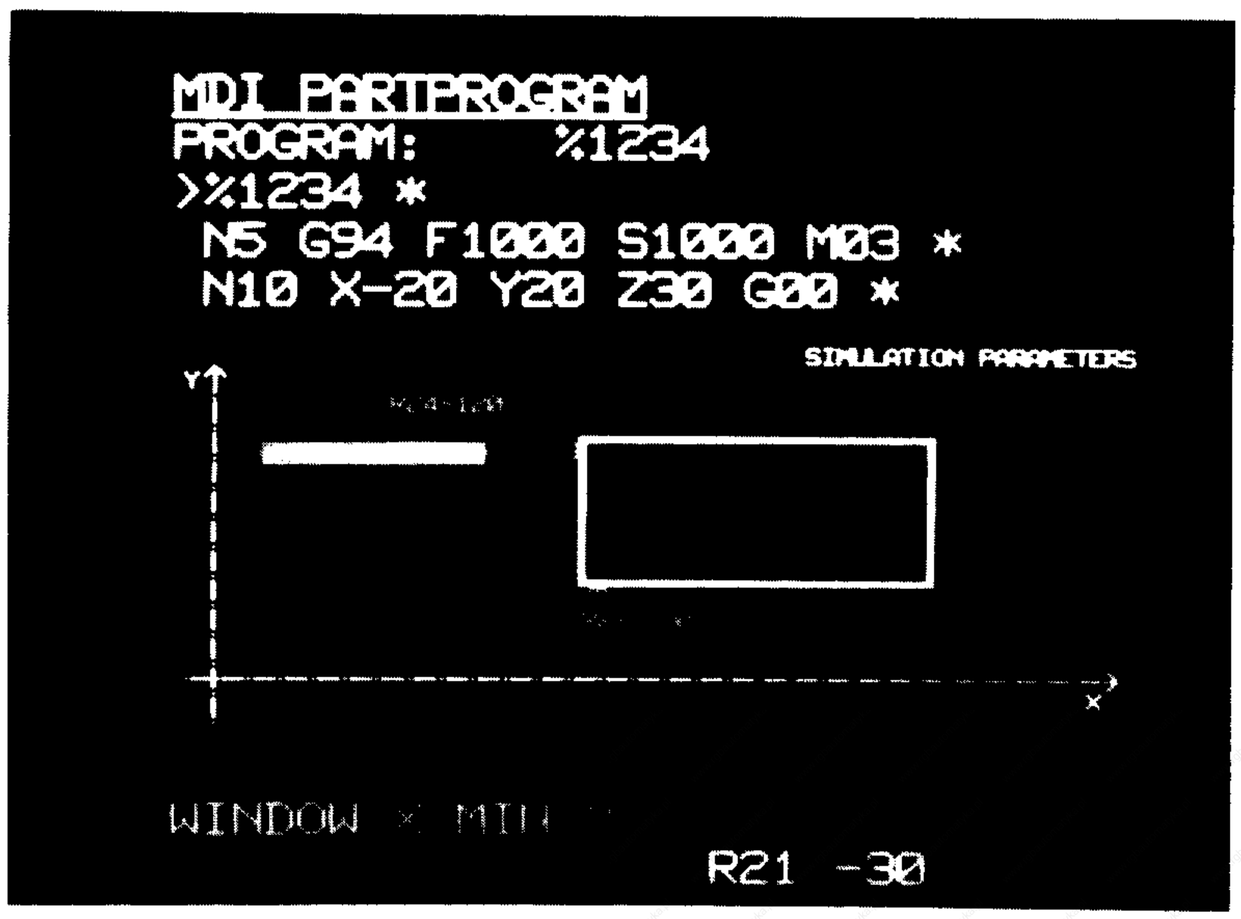


R parameter	Texts for operator guidance	Input format	Preset
R13	Blank X min	+/-5.3	0
R14	Blank X max	+/-5.3	0
R15	Blank Y min	+/-5.3	0
R16	Blank Y max	+/-5.3	0
R17	Blank Z min	+/-5.3	0
R18	Blank Z max	+/-5.3	0

Workpiece Dimensioning R13 to R18

For dimensioning, the smallest and greatest values in relation to the workpiece zero are input. Both positive and negative values can be preset depending on the zero point. The "right-hand parameter" must always be greater than the "left-hand parameter".

Simulation Area



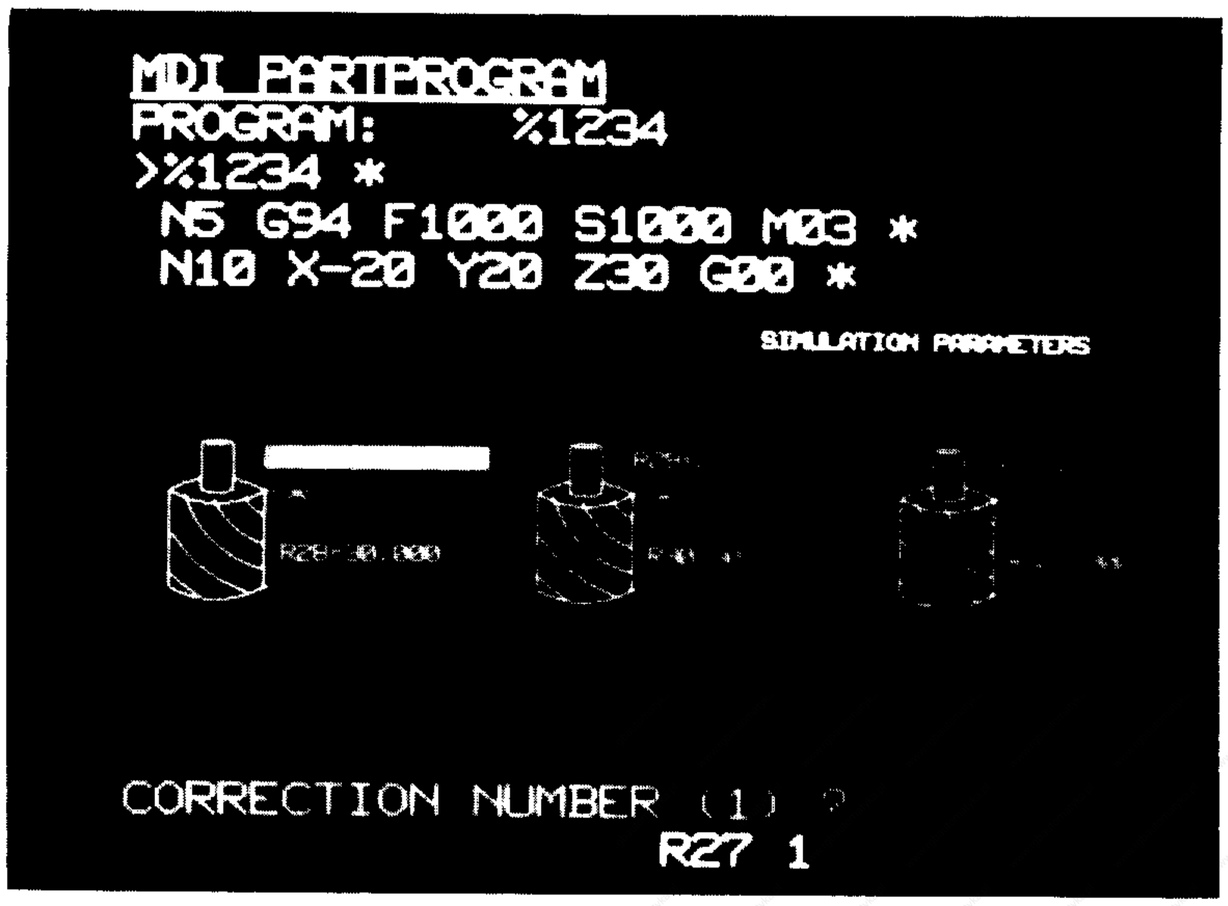
R parameter	Texts for operator guidance	Input format	Preset
R21	Range X min	+/-5.3	Old value
R22	Range Y min	+/-5.3	Old value
R23	Range Z min	+/-5.3	Old value
R24	Range X max	+/-5.3	Old value
R25	Range Y max	+/-5.3	Old value
R26	Range Z max	+/-5.3	Old value

Dimensioning of the Simulation Window: R21 to R26

For the simulation window, the left lower point and the right top point of the respective plane are input. The dimensions are related to the workpiece zero. It is possible to input positive or negative values; the right-hand figure must, however, be always greater than the left-hand figure.

The value range corresponds to the possible traversing path of the control ($\pm$99 m).

The representation of the simulation range defined is optimized by the control in such a way that the greatest value in the X, Y or Z directions can be represented over the entire display. The smaller value is adapted accordingly. The adaptation is based on a side relation of height: width of about 2 : 1.



R parameter	Texts for operator guidance	Input format	Preset
R27	Tool offset (1)	+2.0	Old value
R28	Cutter length (1)	+/-3.3	Old value
R29	Tool offset (2)	+2.0	Old value
R30	Cutter length (2)	+/-3.3	Old value
R31	Tool offset (3)	+2.0	Old value
R32	Cutter length (3)	+/-3.3	Old value

Effective Tool Length: R27 to R32

In the NC tool memory, the radius and length are stored for all tools in relation to the tool reference point. However, since the cutter does not cut along the total cutter length, the tool memory is extended by the effective cutter length for simulation.

The cutter length is the dimension of the cutter which engages for stock removal. However a value can only be entered for a maximum of three offset numbers.

11.4.4 Simulation of an Existing Part Program

1. Set the mode selector at "MDI-PP".

2. Select the "HAUPTPROGRAMM-ANWAHL/SELECTION OF MAIN PROGRAM" or "UNTERPROGRAMM-ANWAHL/SELECTION OF SUBROUTINE" menu by using the "NO" key.

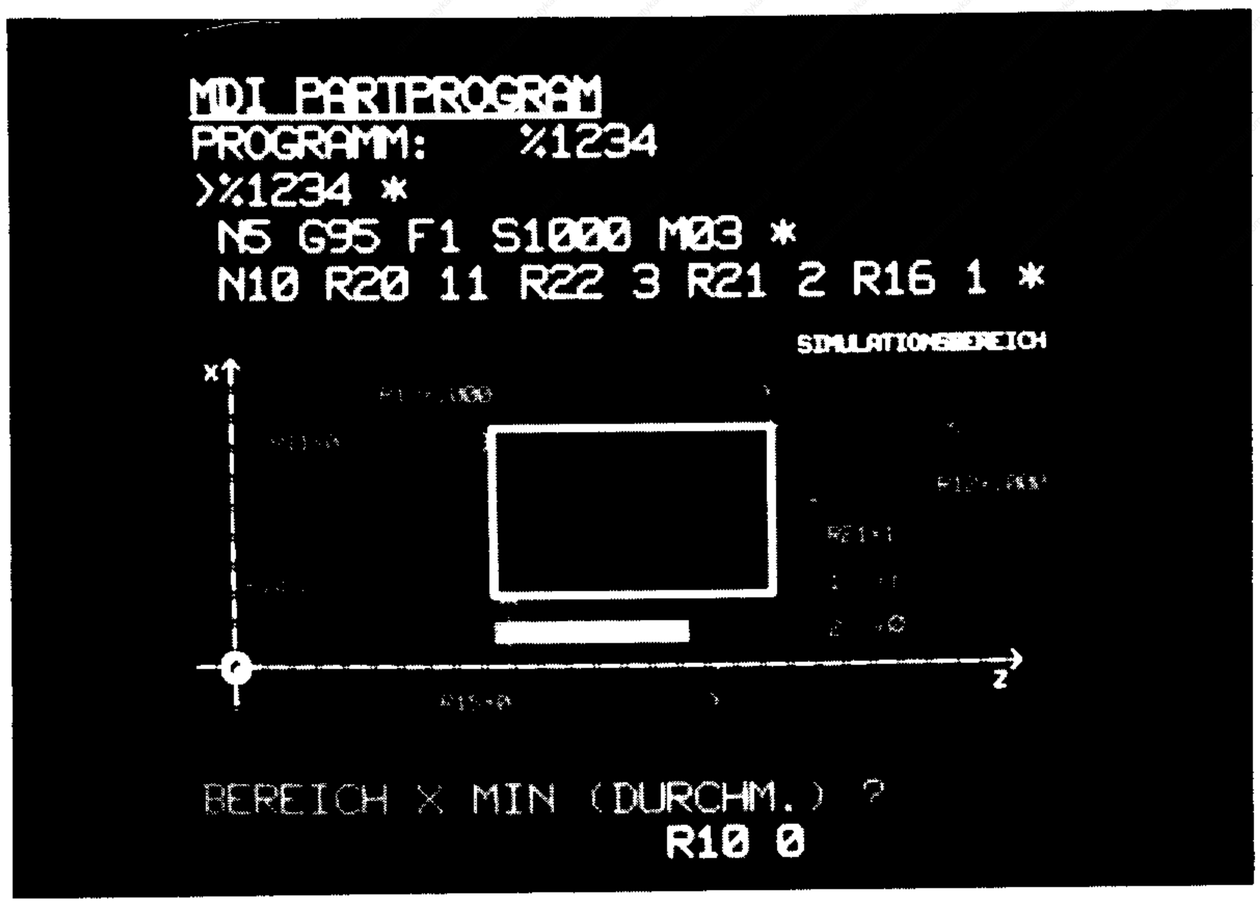
3. Select the part program to be simulated by using the "YES" key. If required, overwrite the offered part program number and then confirm with "YES".

4. Answer the question "PROGRAMMEINGABE ODER KORREKTUR/PROGRAM INPUT OR COMPENSATION" by pressing the "NO" key.

5. Select the "PROGRAMMSIMULATION/PROGRAM SIMULATION" menu by pressing the "YES" key.

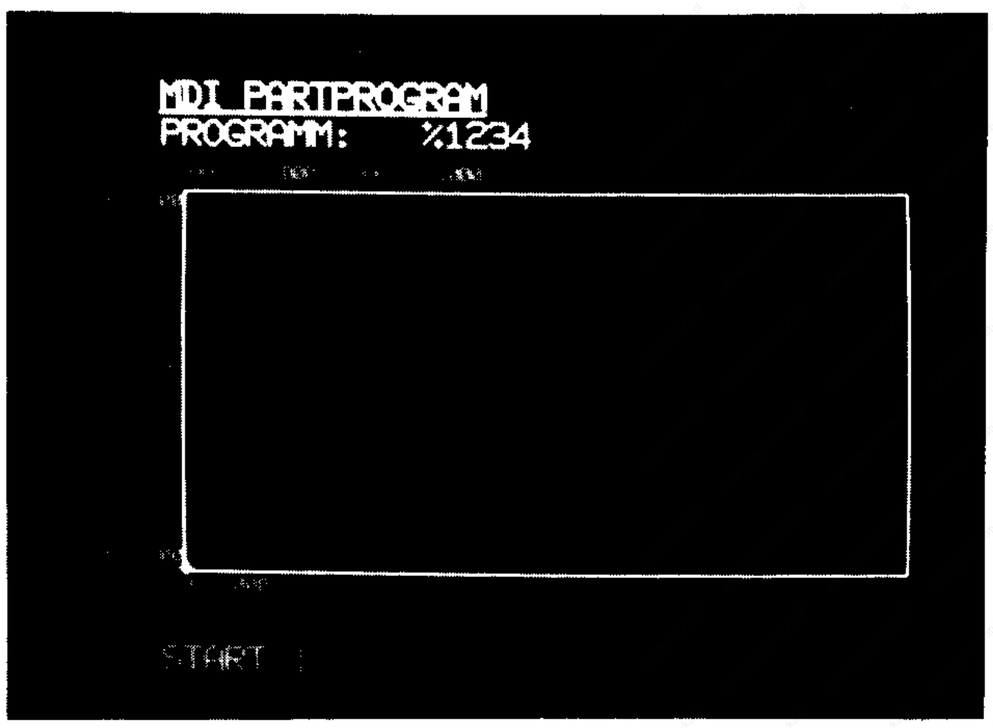
6. If the part program is simulated for the first time, the simulation parameters must be defined. In this case, select the offered "SIMULATIONSBEREICH/SIMULATION AREA" menu with the "YES" key, or proceed by pressing the "NO" key.

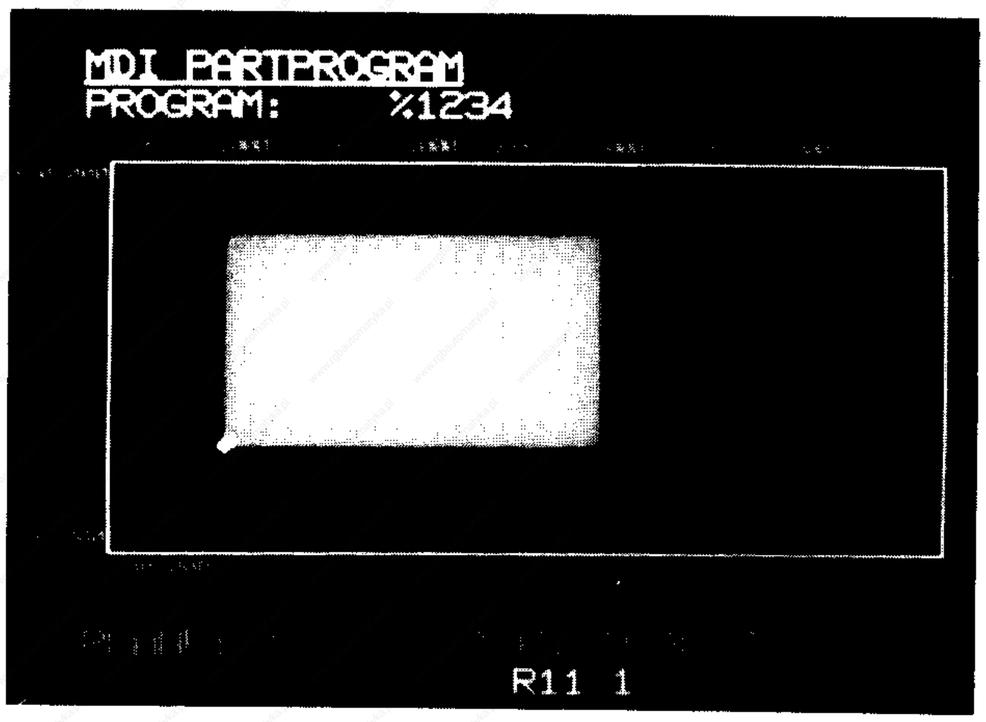
After pressing the "YES" key, one of the following basic simulation displays appears:

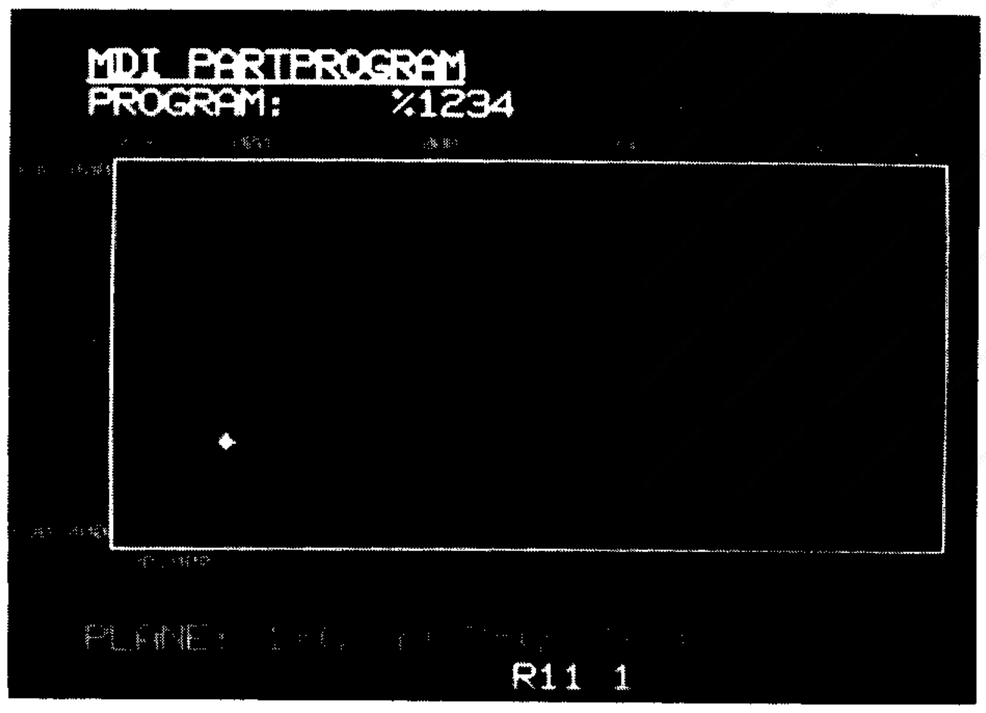


7. In the respective basic simulation display (3T) or series of displays (3M), input the required simulation parameters using operator guidance (see Section 11.4.3.1 and 11.4.3.2) or confirm the existing values by pressing YES or

8. If all parameters are correctly input, answer the question "PARAMETER VALUES CORRECT?" with YES and the simulation display is drawn to the correct scale.
According to the type of representation selected for the 3M control, the blank is displayed on colour background or only in profile.







9. By using the "YES" and "START" keys, graphic simulation of the part program may now be started.

11.4.5 Simulation from a certain Program Position

If longer programs are to be simulated starting at a certain block number, use the "SATZVORLAUF/BLOCK SEARCH" function.

1. Set the mode selector to "Automatic".

2. Perform the block search until the desired part program block is reached (see Operating Instructions, Part 3).

3. Set the mode selector to "MDI-PP".

4. Start the graphics simulation by means of operator guidance. This function starts with the preset block. All previous movements in the program are ignored.

At the end of a simulation, a re-start is possible (SIMULATION NEUSTARTEN?/ RESTART OF SIMULATION?).

If another detail is to be displayed, e.g. in order to recognize details better, it is necessary to redefine the simulation parameters and to restart the simulation.

11.4.6 Abortion of Simulation

11.4.6.1 Abortion of Simulation by Changing the Mode of Operation

A running simulation can be terminated by changing the mode of operation; after that it may not be continued from this position.

11.4.6.2 Abortion of Simulation by Reset

A running simulation may be terminated by pressing the "Reset" key; the basic menu returns.

11.4.6.3 Abortion of Simulation by Program Error

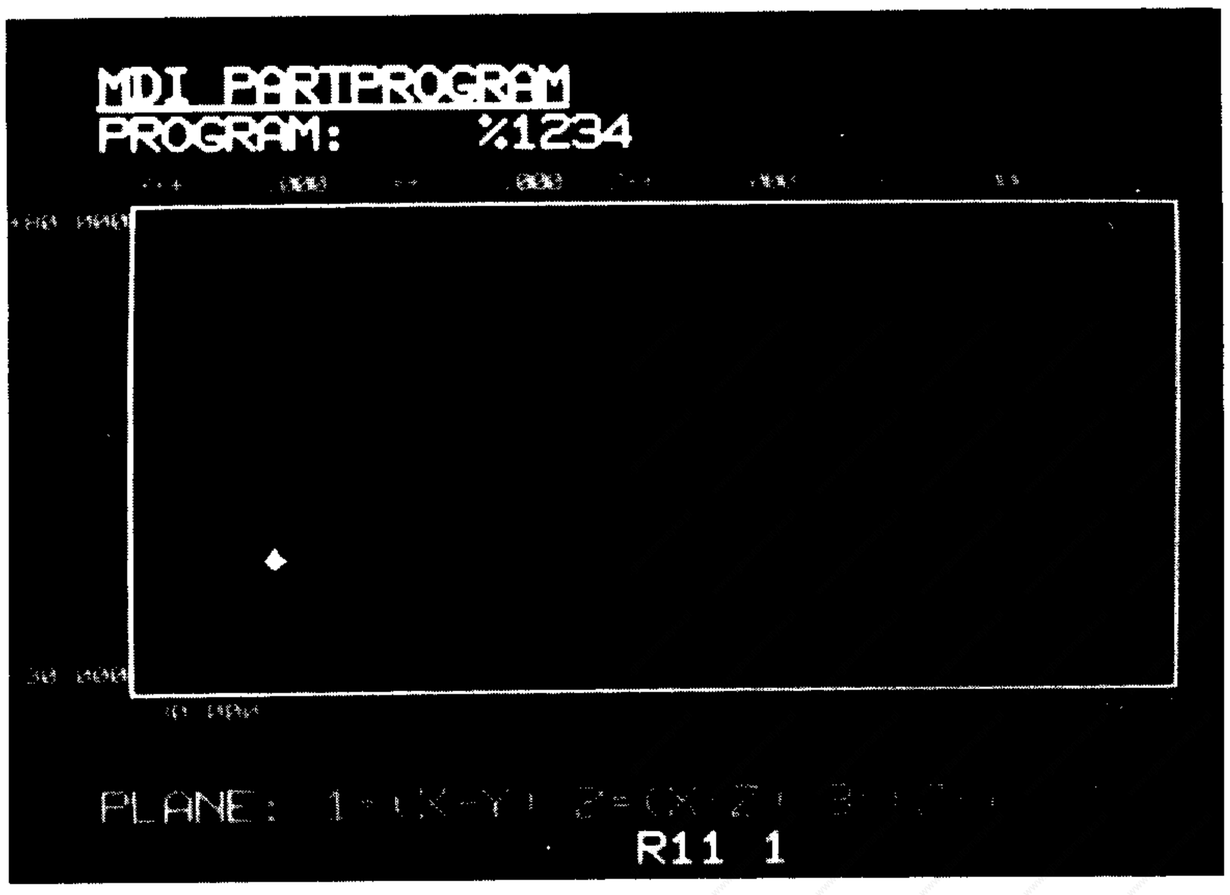
If revoverable programming errors occur during the simulation, they may be displayed by changing the mode of operation to automatic and selecting the correction block.

Restart the simulation after correcting the error (see Section 11.4.4 and 11.4.5).

11.4.7 Change Simulation Plane (3M only)

The selected simulation plane can be changed if the program simulation has not yet been started.

Simulation basic display in X-Y plane with blank.

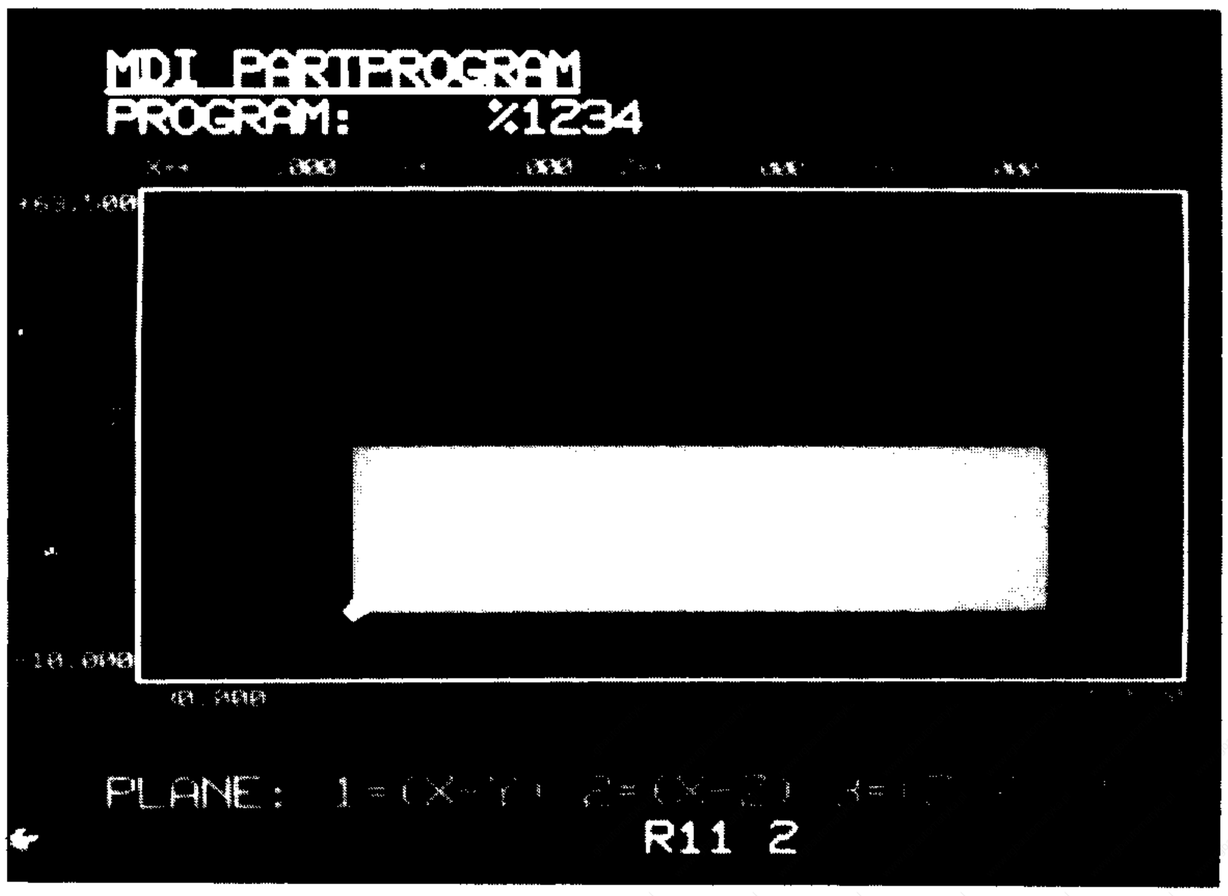


R11 1 Simulation plane is X-Y plane
(see also simulation basic display)
By inputting a new value for parameter R11, the simulation plane may be changed.

2

R11 2 Input simulation plane X-Z and confirm with YES

The simulation basic display now appears with the blank in the Z-Y plane.



R11 3

YES

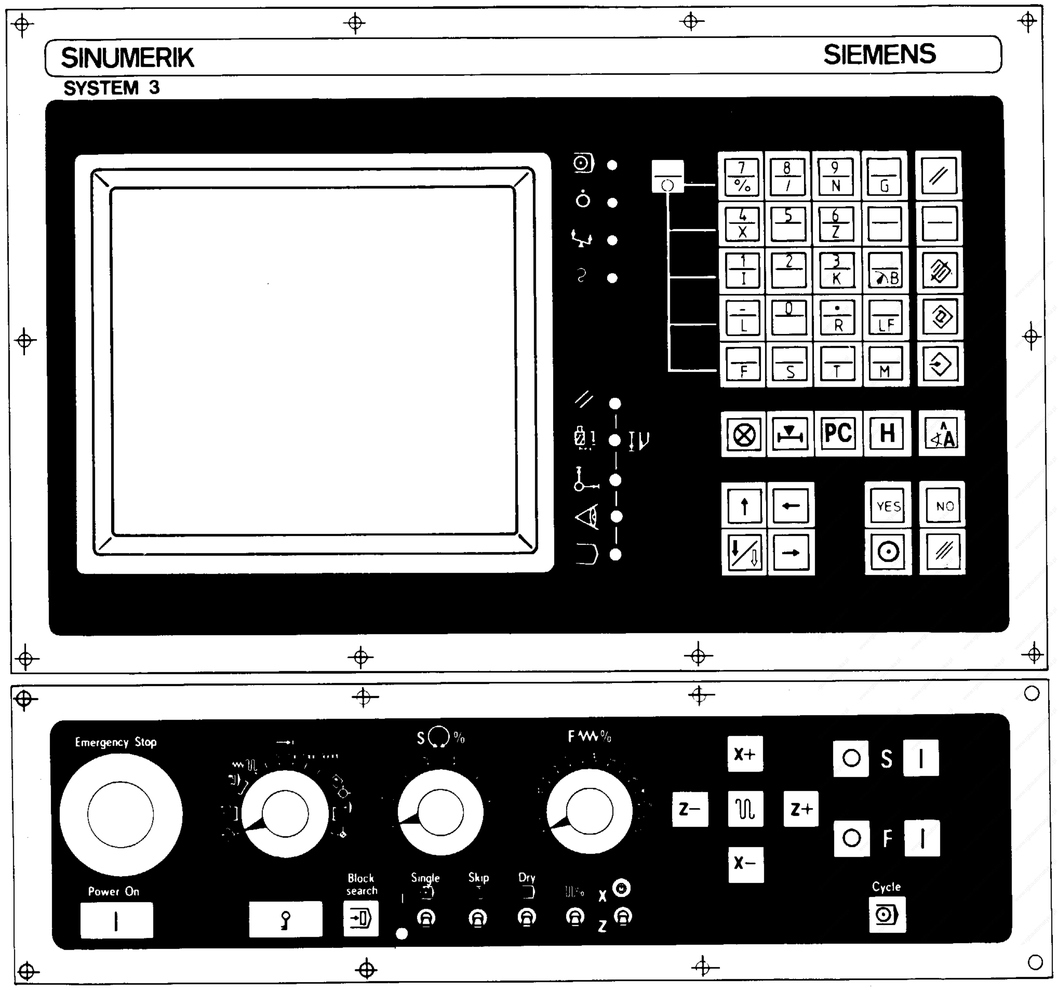
Confirm the selected simulation plane or input another simulation plane.

with

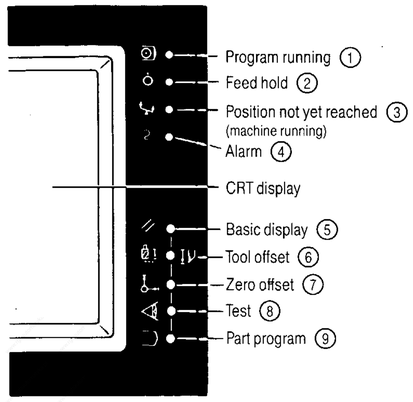
Start program simulation.

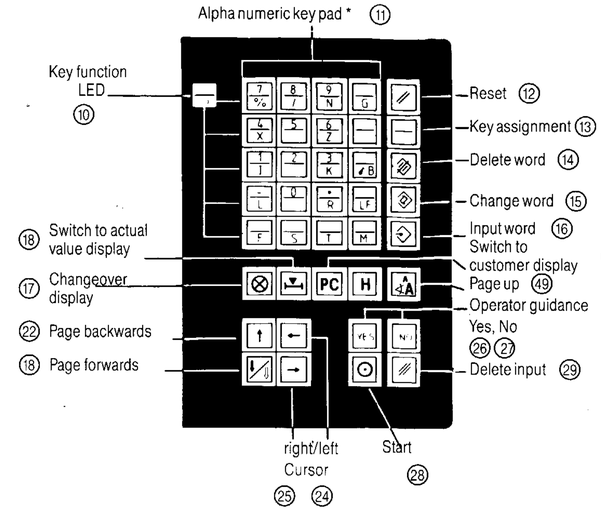
12. Appendix

12.1. Operator Panel and Machine Control Panel (3T)

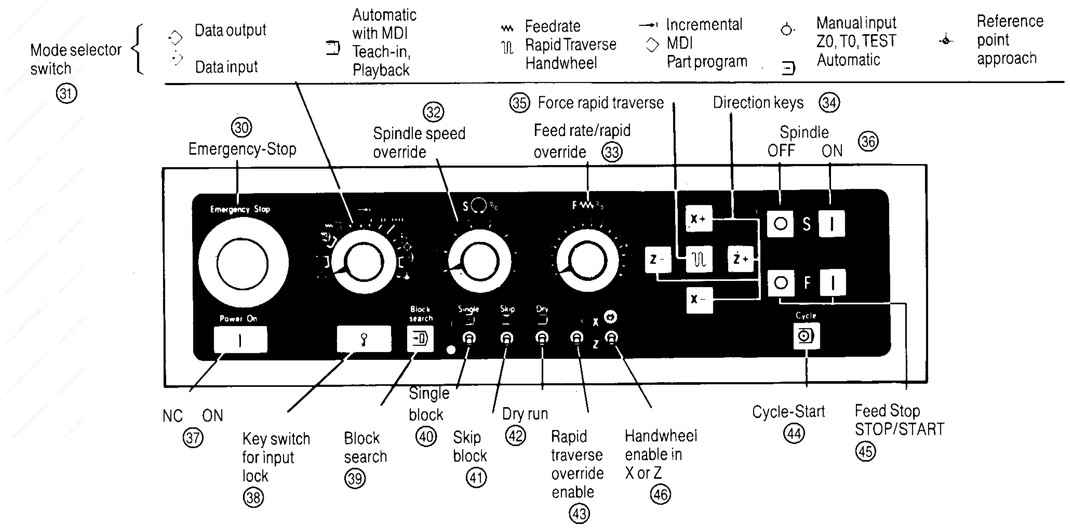


Operator Panel



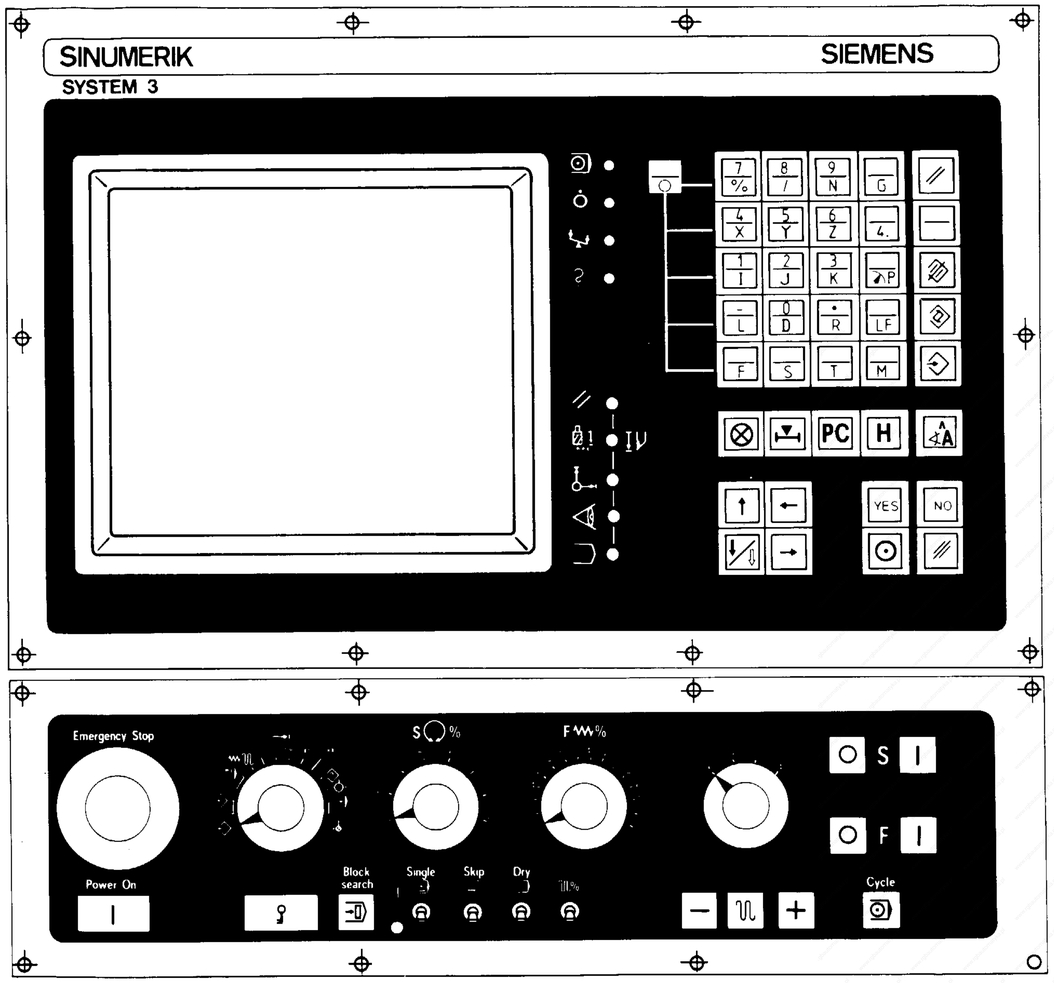


Machine Control Panel

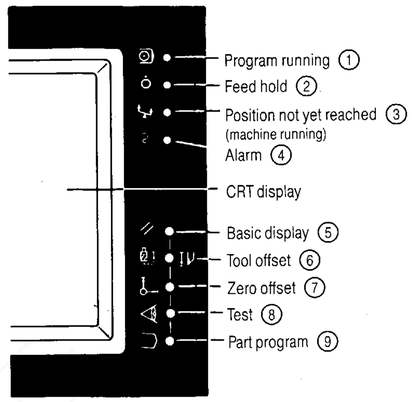


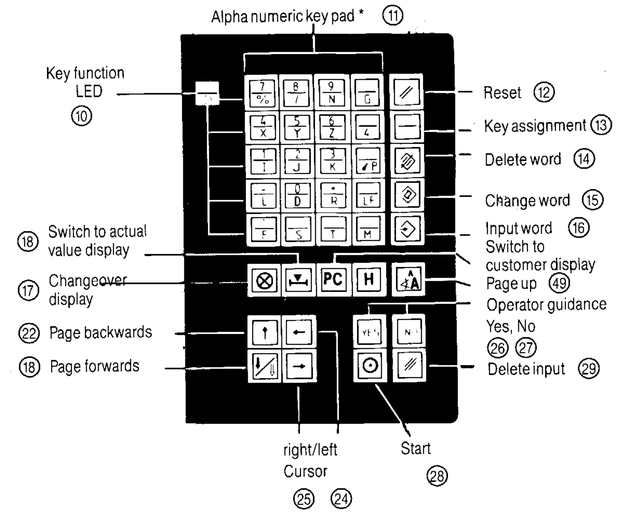
* The special character @ can be entered with double keying, in the sequence %. (percent followed by decimal point).
The 3T with C axis has additional C and J address key functions.

12.2 Operator Panel and Machine Control Panel (3M)

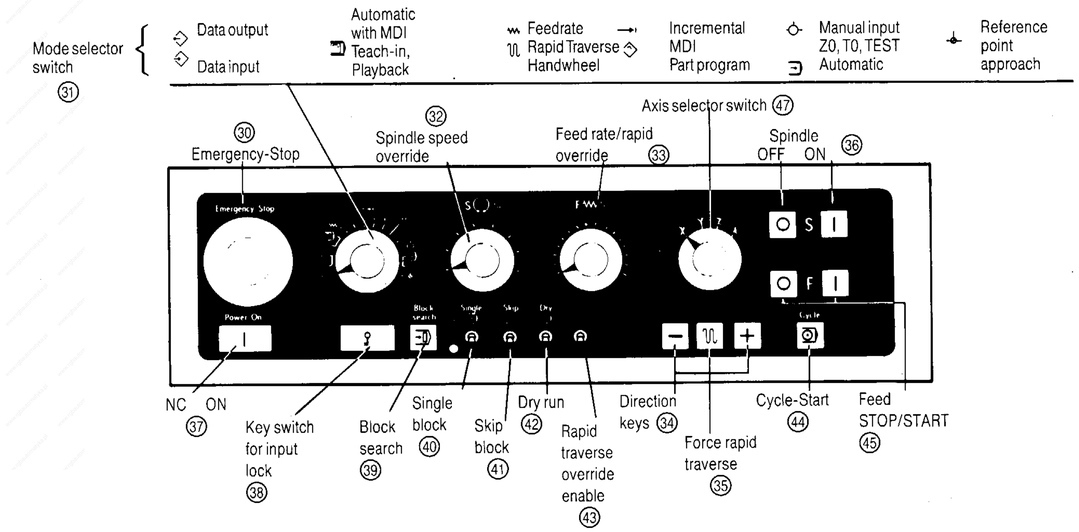


Operator Panel 3 M





Machine Control Panel



* The special character @ can be entered with double keying, in the sequence %. (percent followed by decimal point).

12.3 Abbreviations, Operator Panel and Machine Control Panel Texts

BLOCK SEARCH	
Clear	Cancel input
Cursor	Correction pointer
Cycle Start	NC start
DATA INPUT	
DATA OUTPUT	
DRY	Dry run
Emergency Stop	
INC FEED	Incremental Feed
JOG	Feedrate/Rapid Traverse jog,
	Handwheel jog
M	Feedrate in mm/min
MDA	Automatic manual data input
MDI SE-TE	Manual data input
Page	Page (forwards, backwards)
PART PROGRAM	
PLC	Programmable Logic Control (interface control)
PLC FAULT	Fault in PLC
Power On	Control ON
R	Feedrate in mm/rev.
REFERENCE POINT	Reference point approach
Single	Single Block
Skip	Skip block

12.4 Operator Data

Setting Data NUMBER 25

N	Bit = 0	Bit = 1
0		
0	Automatic block number OFF	Automatic block number ON
0	Punching in ISO code	Punching in EIA code
0	RS 232 (V.24) time monitoring ON	RS 232 (V.24) time monitoring OFF
0	Program start with %	Program start with LF
0	Punched tape block parity OFF	Punched tape block parity ON
0	Operator prompting OFF	Operator prompting ON
0	Display of machine-related actual value system	Display of workpiece-related actual value system

Setting Data NUMBER 26

N	Bit = 0		Bit = 1	
0				
0				
0				
0				
0	Imaginary cartesian coordinate system		Real machine system	
0	Handwheel pulse evaluation increment per division	0	Handwheel pulse evaluation increment per division	100
0	Handwheel pulse evaluation increment per division	0	Handwheel pulse evaluation increment per division	10
0	Handwheel pulse evaluation increment per division	0	Handwheel pulse evaluation increment per division	1

12.5 Alarm List

Clear	No: 3rd Decade / 1st + 2nd Decade	8 Bit 7	7 Bit 6	6 Bit 5	5 Bit 4	4 Bit 3	3 Bit 2	2 Bit 1	1 Bit 0
	00	Axis 1			Axis 1			Axis 1	Axis 1
	01	Axis 2			Axis 2			Axis 2	Axis 2
	02	Axis 3			Axis 3			Axis 3	Axis 3
	03	Axis 4			Axis 4			Axis 4	Axis 4
	10	Axis 1			Axis 1	Axis 1	Axis 1	Axis 1	Axis 1
	11	Axis 2			Axis 2	Axis 2	Axis 2	Axis 2	Axis 2
	12	Axis 3			Axis 3	Axis 3	Axis 3	Axis 3	Axis 3
	13	Axis 4			Axis 4	Axis 4	Axis 4	Axis 4	Axis 4
	22	Spindle encoder contamination error			Spindle speed too high	Spindle measuring loop *) monitor	E-Stop	Control not ready	
	23	Time monitor RS232 (V.24) interface	Overflow 2 Reader Hardware error	Step lock error	Overflow	Parity	Control word overwritten	Overflow 1	Step lock parity error
	24							Over-temperature	
	25		Block w/o LF, or block) 120 characters			Operator error RS232 (V.24) interface	Parity error in memory	Program not available in memory	Block not available in memory
	26						G35 F too large		No coincidence found in block search
	27	Memory overflow	Stored prog. ≠ tape prog.	Tape format error	Tape input locked	Block with more than 120 characters	Block parity error	Irrelevant EIA character	Character parity error
	28	Subroutine error	Intersection error						General decoding error
	29		Wrong input parameter	Wrong block sequence	Incorrect G02/G03	Incorrect radius value	Incorrect angle value	No intersection	Incorrect input value

RS232 (V.24) alarms

Contour errors

*) cleared only with PRESET
(switching the control OFF and ON again)

① = Setting ref. value not possible ② = Drive enable inhibited for moving axis ③ = Limit switch ⊖ ④ = Limit switch ⊕ ⑤ = Measuring System contamination error
⑥ = Drift too high ⑦ = Control loop Hardware ⑧ = Contour monitor ⑨ = Set speed too high ⑩ = Standstill monitor

Alarm List (Continued)

Clear **	No: 3rd Decade / 1st + 2nd Decade	8 Bit 7	7 Bit 6	6 Bit 5	5 Bit 4	4 Bit 3	3 Bit 2	2 Bit 1	1 Bit 0
	30	Circle end-point error	No half for full degree, Pos., rotary axis			Illegal value for ZO or TO	G92P incorrect	Option not available	Circle not in the selected plane
	31	Axis addition not poss.	Too many axes to be moved	No F-word programmed or too high		Thread lead incorrectly programmed	Thread increment too high		
//	32							Inadmissible block with CRC	
	33								
	34								
	35								NC start w/o reference point
	50	Axis 2x or more than 2 axes programmed		CRC contour error	Blueprint programming error	Incorrect block structure	← Reparable Prog. errors →	More than 6 geometry parameters	General decoding error
	51	Software prelimit switch over-travelled	DVC not possible (level-up thread)	Whole block cannot be displayed	Defined block not available	Block with more than 120 characters	Memory overflow	Input locked	Input only in Reset condition
//	52	KV-factors not defined	Axes KV-factors unequal	← Contour monitor error →	Part program cannot be changed		Stop at threading		Strobe input error
	53	General input error	Last prog. not terminated	Only 2 axes permitted in Playback	Playback allowed only when axes stationary	Playback only OK when MDA interrupt	Playback not allowed as 1st block	Prog. No. already used	Block with more than 40 characters
	54	← Contour monitor error →							
	55							Incorrect input parameter	Incorrect input value
Rerun	70	← Graphics alarms →						Incorrect address code in machine datum	
	71								Battery alarm

** The input line (bottom) must be completely cleared.
Error 70X cannot be cancelled with either RESET or CLEAR.
Error 71X can be cancelled with CLEAR.

12.6 Alarm Description

Alarm Number	Description and Remedy
1, 2, 11, 12, 21, 22, 31, 32	Limit Switch The alarms are also triggered when a software limit switch has been approached (value set under TEST N160-N173) (input dependent on the reference point). The axis is stopped, but the position control loop remains closed; the following error is worked in. The axis must be moved in the opposite direction in jog mode, and the alarm must be cleared with the red Reset key.
5, 15, 25, 35	Drive Inhibit of a Moving Axis This alarm is triggered when, the "Drive Enable" signal is cancelled during a traversing movement in automatic mode. It results in a rapid braking of all axes, and the control loop is blocked (Emergency Stop state). A check has to be made to see why the drive enable was cancelled by the interface control (for interface tests, see the Commissioning Instructions). The alarm is cleared with the red Reset key. After this, a program restart is necessary.
8, 18, 28, 38	Reference dimension Setting Not Possible This alarm is displayed, if one of the following conditions is missing: - Reference point approach mode selected - Axis in exact positioning - Traversing command

Alarm Number	Description and Remedy
101, 111, 121, 131,	Standstill Monitoring Clamping errors occur when the defined "Clamp Tolerance-Position Monitor at Standstill" limit is exceeded, or whenever the axis does not arrive at position within a predetermined time. Possible causes. - An axis which should be at standstill is pushed out of position because of high mechanical forces or faults in the servo unit, tacho generator, motor, mechanical parts or in the position control loop hardware. - An axis does not reach position because, for example the drift is too high. - A mechanically clamped axis is pushed out of position. - Other, see Commissioning Instructions. Remedy - The clamp tolerance limit TEST N110-N113 must be higher than the exact stop limit TEST N100-N103. - The "Dwell Time Position Monitor" TEST N353 must be large enough, for the following error to be eliminated within the given time. - Another traversing axis is pushing the axis out of position by more than the tolerance defined in TEST N110-N113 (clamping).

Alarm Number	Description and Remedy
102, 112, 122, 132	Set Speed too High This alarm is displayed when the set speed generated internally by the control is higher than the "Set Speed Limitation" defined TEST N354,. It can be triggered if, for example, the motor cannot follow the set speed (the maximum permissible following error is exceeded). Check that the TEST N354 value is approximately 20% higher than the "Set Speed Limitation" given in N140-N143.
103, 113, 123, 133	Contour Monitoring If this monitor is triggered, alarms 103...133 appear and the drives are stopped by switching the set speed to zero. In addition, the servo drives are inhibited and the control is switched into follow-up mode. The alarms are cleared with Reset. The triggering of alarms 103 through 133 indicates that the speed control loop is poorly optimized, or that the K_v factor is too high for this particular machine. This can be true, even if the value for TEST N351 and TEST N352 is zero. Alarms 103...133 are triggered when the fixed tolerance band N352 is exceeded. This occurs when the axis does not reach the new velocity within the time limit determined by the K_v factor during acceleration or deceleration of the drives.

Alarm Number	Description and Remedy
104 114, 124, 134	Position Control Loop Hardware The monitor is triggered when control-loop cables break when there is a short circuit, and if position control signals are missing. Activation of the hardware monitor on the position control boards causes an immediate stop of all axes (see the Commissioning Instructions). Check the position control loop cables.
105, 115, 125, 135	Drift too High The control is capable compensating for drift (temperature influences on components) in the position control loop (see Section 8.6.5). This alarm is displayed if the drift exceeds approx. 500 mV. The "Position not yet reached" LED does not go out if, for example, the position control loop or the drive are not ready, if the drive is inhibited, if there are hardware malfunctions in the position control loop or drive unit, or if the drift was not properly set on the drive unit. Make a new drift adjustment (Commissioning Instructions) Check the drift values in TEST N230-N233.

Alarm Number	Description and Remedy
108, 118, 128, 138	Measuring system If a linear scale system is used, the EXE issues a hardware signal to the NC. (See Interface Description, Section 7.) Only alarm message; (no effect on automatic mode).
222	Position Control Loop Not Ready (Position Control Input Signal) This indicates a malfunction in the control. It is displayed when the corresponding "Speed Control Ready" input signal (group signal for all axes) is connected, and a control unit is defective (e.g. fuse blown, overheating, etc.). The alarm causes rapid braking of the feed drives and the "Position Control Loop Ready" signal disappears. If this signal is not used in the interface control insert jumper P - N "Servo Ready Simulation" (see Commissioning Instructions).
223	Emergency Stop This alarm is indicated when the interface generates the E-Stop interface input signal. Check whether the E-Stop key has been activated, or if the axis has moved to an Emergency Stop Cam. (Interface test No. 8, byte 5, bit 7 = "0" * E-Stop signal is present). The alarm causes stopping of the axes and drive inhibition.

Alarm Number	Description and Remedy
224	Position Control Monitor Spindle The hardware monitor of the spindle is active only if N407, bit 2 = 1 (pulse encoder present). The alarm brings the spindle to standstill.
	For description, see Alarms 104, 114, 124 and 134.
225	Spindle Speed too high The max. spindle speed (smallest value of selected gear stage, machine data 370 or G92 S... with V-constant) including the tolerance band (machine data 367) was exceeded. Alarm message 225 can now be inhibited via machine datum MD 428, bit 4. If this bit is active, only two situations are monitored: - maximum spindle speed - limitation G925 with constant cutting speed G96. In this way it is possible to change gear without stopping the spindle.
228	Contamination error pulse encoder spindle Only for encoder with EXE 600/601 (linear system) Alarm 108...138

Alarm Number	Description and Remedy
	RS232 (V.24) Alarms
231	Stop Bit Error, Parity Error This alarm is only applies to the Siemens tape reader. - Machine data - Reader
232	Overrun 1 This alarm is only applies to the Siemens tape reader. MS 600 PCB in the reader
233	Control Word Overwrite This alarm is only applies to the Siemens tape reader. - MS 600 PCB in the reader

Alarm Number	Description and Remedy
234	Parity Error Only active when bit 4 of machine data 411 or 412 is set. The alarm is set when the data word from the reader (8 bit information + 1 parity bit) has the wrong parity. This error has nothing to do with parity errors of ISO or EIA tape characters (see alarm 271). - Check machine data setting and reader.
235	Overrun Error This alarm is set when the control has not read (stored) a character before the next character is transmitted. - Check machine data and reader - Error in the USART-interface - Cable
236	Stop Bit Error This alarm is set when the wrong number of stop bits are set. - Check machine data setting and reader.

Alarm Number	Description and Remedy
237	Overrun 2 This alarm only applies to the Siemens tape reader. - Reader - MS 600 PCB in the reader
238	Time Monitoring RS232 (V.24) Interface This alarm is set when the NC does not send or receive data within a period of 20 sec. Cause: - External device not switched on - Wrong cable - External device blocks the CTS signal longer than 20 sec. - If the control signals (DC1-DC4) are used and the NC in data output mode does not receive the DC1 (11H) signal within 20 sec.
242	Overtemperature This alarm appears if the temperature of the components reaches the temperature limit range. (59.7 ^{o}C). When this happens, the "NC Ready 1" signal disappears. The interface inhibits the Read Enable, i.e. only the currently active block is processed. Check the fan and the air flow. If the temperature inside the control is lower than 59.7^{o}C, check the temperature switch S1 on the board 03840.

Alarm Number	Description and Remedy
251	Block Not in Memory e.g. when jumping to a block number
252	Program Not in Memory The preselected part program is not stored in the memory.
253	Parity Error in Memory
254	Operator Error RS232C (V.24) Interface - NC "data output" mode and data start from PLC - RS232C (V.24) inhibit active with data start from the PLC or operator panel. - The coding for the Siemens reader is set in machine data 412.
257	Block without LF, or Block with more than 120 Characters or M02, M30 without LF
261	No Coincidence Found during SNS The alarm appears if, during block search, the desired block or subroutine could not be found before the end of the program, i.e. the desired block or subroutine is not in memory.
263	G35 F too large The F value was selected in such as way so that the thread lead becomes Ø. The block end is not reached, since the calculated feedrate is Ø.

Alarm Number	Description and Remedy
271	Character Parity Error The controller automatically defines the code in ISO or EIA depending on whether the program start definition character in "%" or "EOR". All subsequent characters are checked for parity; if their parity does not agree with the parity initially established, alarm 271 is triggered.
272	Inadmissible Hole Combination of an EIA Character If a character read in is not defined in the EIA code, despite its correct parity, the alarm is activated.
273	Block Parity Error If the block parity monitor (setting data) is active, all the characters of a block are counted. Alarm 273 is signalled if an odd number is counted. The control always punches the tapes with even block parity, irrespective of the setting datum; it fills in blanks if necessary.
274	Block with more than 120 Characters Alarm 274 is activated if a block with more than 120 characters is read. Only stored characters are counted, i.e. carriage returns, sprocket holes, blanks outside comments are not counted. - Split the block into several blocks

Alarm Number	Description and Remedy
275	Tape Input Locked This alarm is triggered if: a) the keyswitch is in the OFF position, when the machine datum setting is "Keyswitch active during part program entry" during the reading of part programs or subroutines, b) the data protection switch on link module 03 840 is not in the "free" position (upper position) during the reading of machine data "TE".
276	Tape Format Error Alarm 276 is signalled if: a) the permissible number of decades after an address is wrong, b) a decimal point appears in a wrong position, c) part programs or subroutines are not defined properly, or are terminated. d) the format is incorrect when deleting programs
277	Stored Program ≠ Tape Program If a tape is read in more than once, its contents are compared block-by-block with the program stored during the first reading. If any discrepancies are noted, alarm 277 is signalled. This alarm is also most frequently triggered when an attempt is made to store a program under the same program number as that of a program already in memory. This already stored program must then be deleted. In contrast to storing, the "FREE MEMORY" remains unchanged when comparing.

Alarm Number	Description and Remedy
278	Memory Overflow If the memory space becomes insufficient during reading, alarm 278 appears. The amount of memory still available can be checked with the "Free Memory" number. If necessary, programs not required must be deleted and the program read in anew.
	Irrecoverable Programming Errors
281	General Checking the faulty block in the "Correction block display"
287	Intersection with @ 22 Alarm is indicated if the L94 stock removal cycle has been programmed with wrongly defined parameters.
288	Subroutine Error - M17 in the main program - Nesting depth exceeded
	Errors in Blueprint Programming
291	False Input Value The values programmed lead to overflow during calculation or do not allow calculations because of wrong dimensions or procedures or the radius being too large.

Alarm Number	Description and Remedy
292	<u>No Intersection</u> When the elements of the described contour are calculated, the programmed values do not result in an intersection.
293	<u>Wrong Angle Value</u> Angles equal to, or greater than 360^{o} do not make sense in terms of the described contour.
294	<u>Wrong Radius Value</u> The input value is too large or inadmissible in terms of the described contour.
295	<u>Wrong G02/G03</u> Arc direction not possible for the described contour.
296	<u>Wrong Block Sequence</u> Several blocks are necessary for calculation: the sequence of blocks is not compatible, or there is insufficient information. e.g.: N15 ... B15 LF N20 ... G03 I20 LF
297	<u>Wrong Input Parameter</u> The programmed parameter sequence is inadmissible or incomplete for the described contour. e.g.: N15 ... X60 B15 LF (Z axis missing) N20 ... X90 B10 LF

Alarm Number	Description and Remedy
301	Circle Not in the Selected Plane 3 M: the interpolation parameters are not correct in terms of the selected plane.
302	Option not available Check software complement with machine data.
303	G92 P Incorrect - Working diameter to resolution diameter factor = 0 - Using illegal address characters.
304	Zero Offset or Tool Offset: Inadmissible Value With values of six or more decades, double word overflows are possible.
307	No Half or Full Degree Position With Rotary Axes (Axis with switchable table) The rotary axix can only be positioned in full or half degrees. The programmed position plus possible zero offsets and other compensations does not lead to a full or half degree position.
308	Circle End-point Error "Circle End-point Monitor" (N355) machine datum. The programmed circle end-point is outside the circle. The input limit has been exceeded.
313	Lead change too large The change value programmed with G34/G35 (variable thread) is not in the range 0.001 to 16.

Alarm Number	Description and Remedy
314	Thread Lead Incorrectly Programmed The thread lead is programmed under I, or K, and always refers to the leading axis. If not, alarm 314 is activated. (e.g. X 20 000 Z 10 000 K 1000)
316	No F word is Programmed No F value programmed or F programmed for rev. feed-rate. With the TRANSMIT function, the programmed feed-rate is reduced to such a level as not to exceed the maximum C axis speed.
317	Too Many Axes to be Moved Only appears if too many axes would move because of the G41/G42 function.
318	Axis Addition not Possible An alarm is only displayed if the "TRANSMIT" function has been activated. Linear interpolation: Only one axis of the imaginary plane and further axes outside the imaginary plane have been programmed. Circular interpolation: An imaginary axis has been programmed, but the circular plane is not the imaginary plane. Example: G17 X-Y circular plane X-Z imaginary plane Programmed G17 GO2 X.. fault

Alarm Number	Description and Remedy
322	Inadmissible Block for Cutter Radius Compensation/ Tool Nose Radius Compensation With a selected CRC, G92, G59, G33 may not be programmed. Remedy: Program G40 or G41/G42, D00 (CRC deactivation) or G41/G42, T00 (TNRC deactivation)
351	NC start without Reference Point

Alarm Number	Description and Remedy
	Recoverable Programming Errors
501	**General** The error is indicated in the "Correction Block Display" (the block before the decoding), with an additional character.
502	**More than 6 Geometry Parameters** (programmed in a block) Geometry parameters are: axes, interpolation parameters, radii, angles.
504	**Wrong Block Structure** e.g. N10 G02 X1000 LF (interpolation parameter missing) N20 G02 Z2000 I20 LF (I20 not allowed in this block)
505	**Contouring Error** This alarm is displayed when alarm 302 is active. In this case the "Blueprint programming" option is not available.
506	**CRC/TNRC Contour Error** The intermediate block for the selected offset is too small, or the offset calculations result in traversing a direction opposite to the one programmed.
508	**Axis 2x or too many Axes Programmed**

Alarm Number	Description and Remedy
511	Input only in Reset State This alarm occurs when inadmissible functions are selected in Automatic mode. These functions are only admissible if the Reset key has been activated beforehand. The alarm can be cleared with  .
512	Input Inhibited Inputs are possible only when the key switch is in the "free" position, or if the data protection switch on the 03 840 link module is in "upper" position. The alarm is also displayed if entering in the wrong operating mode.
513	Memory Overflow This alarm signals when the program memory is full. Programs that are not used at the time can be deleted if necessary (see Section 7.1). These operations are only possible if the control is in the Reset state.
514	Block with more than 120 Characters During manual data input (editing), the number of characters in a block is checked before storing. (LF must be character no. 120.) If this alarm is displayed, check the entered LFs. If necessary, the block in question can be subdivided into several smaller blocks.

Alarm Number	Description and Remedy
515	Preselected Block Number Not Present This alarm is displayed if a block number is selected which does not exist in the program. The cursor is set to the program start, and the program is displayed.
516	Block cannot be displayed completely With certain block configurations, it is not possible to display all the characters, in spite of the fact that the allowed block length (120 characters) has not been exceeded. Remedy: By inserting an "LF" it is possible to form two blocks, the second of which has no block number. These two blocks can now be altered to generate an executable program again.
518	Software Limit Switch Overtravelled The software limit switch was approached with a greater feedrate than that set in machine data 346.
521	Strobe Input Error This alarm appears during external data input, if the wrong code is used, if the word is too large, or if machine data words are overwritten without feed stop being present.

Alarm Number	Description and Remedy
523	Stop during Threading This alarm is an indication that the thread may have been damaged following a stop during thread cutting. (Stop in feed/rev., for example, due to removal of axis-specific feed enable)
525	Part program cannot be changed The alarm is output if the part program to be changed is not at the end of the part program memory or is currently being processed in automatic mode.
527	Unequal Position Loop Gains (KV factors) of the Axes Inequality in the KV factors of the axes leads to contour deviations. This alarm is triggered if the measured KV factors of all axes are not equal.
528	KV factors have not been calculated This alarm is a reminder after power-on and machine data changes.

Alarm Number	Description and Remedy
	MDA Alarms
531	Block with more than 40 Characters An input block with fewer input characters is necessary.
532	Program Number already Present The program number entered in MDI is already present in the program memory.
533	Playback Not Allowed as First Block In teach-in, a program number must first be specified and correctly stored.
534	Playback only Allowed if MDA is Interrupted An additional block has to be stored in MDA.
535	Playback only Allowed if the Axes are Stationary If the block is to be stored, no more axis motion may take place.
536	More than 2 Axes in a Block No more than 2 axes can be interpolated.
537	Last Program Not Terminated This alarm is a warning that a new program is being opened before the old program has been terminated in MDA.

Alarm Number	Description and Remedy
538	General Input Error in MDA
551	False Input Value The value in the input line is too large or has the wrong format.
552	False Input Parameter The input value does not fit in the operator prompting sequence or an incorrect parameter has been defined (e.g. max. value smaller than min. value).
702	False Address Code in Machine Datum Machine data have to be changed (name, axis).
711	Battery Alarm Check the voltage of the battery on 03500 (power supply). Change the battery, if necessary. Important: In order to prevent the C-MOS Memory 03210 or 03260 from being erased, it is necessary to change the battery while the control is under power. As the alarm can be deleted with "CLEAR", operation does not have to be interrupted. Caution: - In older hardware versions, POWER ON RESET is necessary - In the control version in which the NC and PLC are both in a single-tier rack, the PLC stops if the battery voltage fails. In this case, the NC cannot display the alarm. This state is indicated on module 03840 by a red LED flashing at a frequency of approx. 5 Hz (processor monitoring). No alarm appears on the display.

To
Siemens AG

E 885
Postfach 48 48
D-8500 Nürnberg 1

Suggestions

Corrections

For Publication/Manual:

SINUMERIK System 3
Basic Version 4C

Operator's Guide

Order No.: 6ZB5 410-1AA02-0BA0
Edition: 08.88

Sender:

Name _____

Company/Dept. _____

Address _____

Telephone /

Should you come across any printing errors when reading this publication, we would ask you to inform us accordingly, using this form. We would also welcome any suggestions you may have in the way of improvement.

Suggestions and/or corrections

Energy and Automation Group
Department for Numerical Controls and
Drives for Machine Tools and Robots

Published by Information and Training Subdivision
Postfach 4848, D-8500 Nürnberg 1
Federal Republic of Germany

Siemens Aktiengesellschaft

Order No. 6ZB5410-1AA02-0BA0
Printed in the Fed. Rep. of Germany
886352 PA 6892.

progress in automation: Siemens

08.88