5. Program Execution
AUTOMATIC operating mode

In the AUTOMATIC operating mode, the control system executes programs sequentially. The program or block number, or the NC word with which you wish to start can also be entered in this mode.

Select operating mode

Press the operating mode key.

The operating mode menu is displayed on the monitor:

Press the AUTOMATIC operating mode softkey.

SELECT. PROGRAM	SELECT. BLOCK	SELECT. DELETION LEVEL
PRESE-LECTION RATE	SELECT. NC-WORD	SPINDLE SELECT
CYCLE TIME		AUTO-MATIC-MODE

PROGRAM: % START BLOCK: N
DELETION LEVELS: /

Select program

Press the SELECT PROGRAM softkey.

INPUT PROGRAM NUMBER OR CONFIRM

Confirm.

The displayed program is selected.

Enter digits.

Confirm.

The entered program is selected.

Automatic operation can now be selected if you wish to start the displayed program with the 1st block without selecting a deletion level. Otherwise: Select Start block/deletion level.

Selecting a start block (search for start block)
If a large NC program (including programs containing C-axis milling) is not to be started at the beginning of the program, the EPL 2 control system offers the option of utilizing the start block search to start the automatic routine at any point whatsoever.

If the automatic routine is started with a certain start block in the middle of the parts program, the control system automatically provides all technology data programmed and required for this block (including exchange and compensation of the correct tool). In other words, the control system behaves as if the program had been executed from the very beginning.

Proceed as follows to define the start block:

	Press the SELECT. BLOCK softkey.	INPUT BLOCK NUMBER OR CONFIRM
	Confirm.	First program block is selected.
	Enter digits.	
	Confirm.	The entered starting block is selected and displayed.

Advice: If the automatic routine is started with a certain start block in the middle of the parts program, the control system produces automatically all programmed technology data for this block (incl. change and calculation of the needed tool); that means the control system acts as if the program had been executed from the very beginning.

Example: The program is to be started with block number N6.

```
N1  G92  X...  Z...  K...  T1
N2  ...
N2  ...
N4  ...
N5  G91  ...
N6  G92  X...  Z...  I...  K...  T3
```

Tool T3 is immediately changed and calculated.
The traverse paths are executed incrementally (G91).

General information

Never start the automatic routine in sub-routines when using the start block search function.

Furthermore, the automatic routine is not to be started during cycles, i.e. do not select the program blocks between cycle initiation and cycle end (G80) inclusive as start blocks. However, starting the program in the NC block in which the cycle is initiated is permitted.

Information for machining

If the function start block search is to be used, it is imperative to program both addresses X and Z of the movement conditions for rapid traverse, straight line and circle in each block, even if they remain unchanged.

Searching for start block in conjunction with C-axis machining

To make certain that the same status is reached when executing the start block as when executing the program from its beginning, all axes are moved to the last target coordinates programmed.

During this process, a certain sequence is followed when traversing the axes, irrespective of the programmed addresses X, Z and C. The two axes traversed first are those whose interpolating relationship (override of a linear movement X or Z with the circular movement C) is defined by the G-function of the C-axis machining. Afterwards the infeed movement to the workpiece is executed.

With the G-functions G100, G101, G102, G103 and G77 therefore, the X-axis and C-axis are first traversed and this is followed by the infeed movement of the Z-axis. With the functions G110, G111, G112, G113 and G78, the Z-axis and C-axis are first traversed and this is followed by the infeed movement of the X-axis. The interpolating axes (X- and C-axis during frontface milling or Z- and C-axis during circumferential milling) always move to their respective target coordinates during the start block search at the rapid feed rate defined under parameters N62 or N126.

The infeed movement is performed at the "feed in minutes" defined under the address F of function G94.

Information: Since the feed values of milling movements are generally very low, time considerations make it advisable to have the traversing movements of infeeding start a short distance from the workpiece.

Example

```
...
...
N50 G94 F40
N60 G1 Z-5
N70 G101 X65 C100
N80 G101 X50 C120   <-------- start block
N90 G101 C210
...
```

After the program start the X-axis and C-axis move to target coordinates X50 and C120 in rapid traverse. After both interpolating axes have reached their target points, infeed to the workpiece (Z-5) is performed at the milling feed of 40 mm/min.

Start block with linear movement

If the start block contains the function G0 or G1, the infeed movement is also performed last in this case too.

Example

```
...
...
N50 G94 F40
N60 G101 X40 C20
N70 G1 Z-8        <-------- start block
N90 G101 C210
...
...
```

After the program start, the X-axis and the C-axis move to target coordinates X40 and C20 in rapid traverse. Once both interpolating axes have reached their target points, infeed to the workpiece is performed at the milling feed of 40 mm/min (last programmed coordinates in Z-direction).

Start block without tool movement

If the start block does not contain a movement function, the first NC block after the start block which does have one is treated as the start block; the same then occurs as described above.

Cycle initiation "Hole circle frontface" as the start block

If the start block contains the hole circle cycle G77, the first step will be that the C-axis will move in rapid traverse to the first angular value programmed under address I in the cycle initiation and the X-axis will be moved to the last valid X-value. Thereafter the feed axis moves in the Z-direction to the start position at the last programmed feed rate.

Cycle initiation "Hole circle circumference" as the start block
If the start block contains the hole circle cycle G78, the C-axis moves in rapid traverse to the first angular value programmed under address I in the cycle initiation, the Z-axis to the last valid Z-value position. The infeed axis then moves in the X-direction at the programmed feed to the start position.

Cycle initiation "Angular circle" as the start block
If the start block contains the angular circle cycle G770, the procedure with this function is the same as in frontface milling, irrespective of whether circumferential or frontface milling is specified in the contour description. This means that the X- and C-axes always move first in rapid traverse, then the Z-axis at the last feed specified. It is not possible to start in the contour description; an attempt to do so triggers an error message.

Start block with zero point shift of the C-axis
If the zero point shift of the C-axis G152 is programmed in the start block, the last valid coordinates are approached. The control system compensates the zero point shift as early as the start block.

Example

```
...
...
N40 G1 Z-2
N41 G101 X60 C20
N42 G152 C45    <-------- start block
N43 G101 X25 C80
...
...
```

After the automatic routine is started, the zero point s hift of 45ø is activated.
The target coordinates X60 and C20 are approached in rapid traverse. Infeeding in the Z-direction (Z-2) at the last active feed then ensues.

General information for C-axis machining

Insofar as the above is concerned, the operator must therefore make certain before actuating the cycle start button that there is no risk of tool and workpiece colliding.

Select deletion level
(see also section 3.5)

Press the SELECT.
DELETION LEVEL softkey.

INPUT DELETION LEVEL:

Confirm.

No deletion level is activated.

Enter digits.

Confirm.

Entered deletion level is activated
and displayed.

Select NC word

Press the SELECT NC-
WORD softkey.

INPUT NEW SEARCH WORD

Enter desired NC search
word. (e.g.: G3, V1, ...)

Confirm.

The first block containing the entered
search word is displayed.

This selection mode need not always be executed completely.
If the desired program has been selected once before and the first block is to be started without activating a deletion level, then it suffices to actuate the AUTOMATIC MODE softkey.

Press the SELECT.
SPINDLE softkey.

The actual value display is switched
over to another spindle (if existing).
(For the program routine without
any meaning)

Quantity preselection
By means of this function, quantity counting can be activated when several identical parts are to be produced.
When working with M99 (end of program with automatic restart), it suffices to preselect the desired quantity. The program will automatically be executed the number of times that was preselected. Then the remaining **rest quantity** is displayed.
When working with M30 (end of program, return to program start), then the program must be restarted after each run by pressing the **CYCLE START** softkey.
Without quantity data, the number of pieces is counted (**quantity**).

Press the PRESELECTION RATE softkey.

INPUT NUMBER OF WORKPIECES:

Enter digits (e.g. 5).

Confirm

5 REST QUANT
0.0 PIECE TIME
0.0 TOTAL TIME

Monitoring cycle time

Switching on or off the "time" display.

Press the CYCLE TIME softkey.

5 REST QUANT
0.0 PIECE TIME
0.0 TOTAL TIME

Starting a program in AUTOMATIC operating mode

Press the AUTOMATIC CONTROL softkey.

AUTO-PARALLEL MODE	AUTO-/ SINGLE BLOCK	SELECT DELETION LEVEL
D TOOL OFFSETS	OPTIONAL STOP	SELECT. SPINDLE
FEEDRATE % ON/OFF	SPEED % ON/OFF	COOLANT ON/OFF

Press CYCLE START.

The selected program is executed.

The following data are displayed in addition to the program and actual value display:
The quantity of workpieces yet to be machined, the momentary machining time of the workpiece and the total machining time. When the display of remaining pieces shows zero, the control system produces "cycle stop" and the message "preselected quantity reached" will be displayed.
From then on, CYCLE START can no longer be activated.
In order to restart automatic operation, you have to quit this operating mode and select AUTOMATIC mode once again.

A release can only be produced by preselecting a new quantity or by confirming the PRESELECTION RATE softkey or by deleting the quantity output. The above mentioned error message will be displayed if the operator attempts to select automatic mode without release.

D-corrections

Values for D-corrections can be entered up to a magnitude of a maximum of 0.999 mm. After the number of the tool to be corrected has been entered, a correction for Z and X can be entered for cutting tools. If the tool is a milling tool, a third correction value DD is also offered for the cutter diameter. The correction of the cutter diameter is not effective until the tool in question is exchanged (or at program start), however, not immediately in the next NC record, since the control's geometry processor has to recalculate the traversing paths.

Feed override

A percentual feed override can be set using the handwheel.
The programmed value is regarded as 100% - the percentage set with the handwheel is carried out.
This value is displayed on the monitor.

Press the FEED % softkey.

F ... 100% (actual value)
F ... NOM. (nominal value)

Handwheel setting F 124%

F ... 124% (actual value)
F ... NOM. (nominal value)

The slides traverse, during execution of the program, at a higher or lower feed depending on the set percentage.

Speed override
A percentual speed override can be set using the handwheel.
The programmed value is regarded as 100 % - the percentage set with the handwheel is carried out. This value is displayed on the screen.

Press the SPINDLE % softkey.

1 S ... 100% (actual value)
S ... NOM (nominal value)

Handwheel setting S 85 %

1 S ... 85% (actual value)
S ... NOM (nominal value)

The spindle turns, during execution of the program, at a higher or lower speed depending on the set percentage.

Switchover:
Automatic/single block

Press the AUTOMATIC/ SINGLE BLOCK softkey.

SINGLE BLOCK

Press CYCLE START.

The program is executed block by block by repeated pressing of the **Cycle-Start key**.

Press the AUTOMATIC/ SINGLE BLOCK softkey.

AUTOMATIC

Press CYCLE START.

The program is executed continuously.

Optional stop

Press the OPTIONAL STOP softkey.

OPTIONAL STOP

The control system interrupts the program at the point where the OPTIONAL STOP function (e.g. for checking workpiece dimensions) is programmed (M1).

Press CYCLE START.

Program execution is continued.

Tool inspection

TOOL INSPECTION softkey

See description of the tool inspection (after SINGLE BLOCK operating mode)

Press "continue" key in the automatic menu.

A follow-up menu appears.

PARALLEL OPERAT. MODES		TOOL INSPEC-TION
		SPINDLE SELECT. <
DISPLAY SELECT	DIAG-NOSIS	

Parallel work in another operating mode
During automatic program execution, you can switch over to **EDITOR**, **DIAGNOSIS** or **PARAMETER** operating mode.
This enables you, for example, to enter a program without interrupting machining.

Press the PARALLEL OPERAT. MODES softkey.

The operating mode menu is displayed.

Press the EDITOR, DIAGNOSIS or PARAMETER softkey.

You will enter the operating mode selected.

The auto-parallel mode is indicated by the inverted colors of the heading of the status line.
In this state, pressing the operating mode key will not terminate automatic operating. The cycle keys are still effective.

Extended actual value display
If in addition to the "large" actual value display for the selected slide the actual values of all spindles are to be monitored, the operator has to switch over to the extended actual value display (see also display configuration in chapter 0)

Press softkey DISPLAY SELECT.

Switch-over to the extended actual value display. The mode of representation corresponds to the values programmed under parameter **N0006.**
If no reference run has yet taken place, an **--- R ---** appears instead of the relevant value.

Press again the softkey DISPLAY SELECT.

Switch-back to the normal actual value display for the selected slide.

Diagnosis as side-line operating mode
If the diagnosis is to be selected as side-line operating mode when in the automatic mode, then proceed as follows:

Press the "continue" key in the automatic mode menu.

The follow-up menu is displayed.

Press the DIAGNOSTOC MODE softkey.

The second main menu of the diagnostic mode is displayed (see also section 7.1 DIAGNOSOS operating mode in side-line operation).

SINGLE BLOCK operating mode

The SINGLE BLOCK operating mode is identical in function and operation to AUTOMATIC operating mode, with the exception that only one block is executed at a time. The next block in each case must be started by renewed pressing of CYCLE START.
This mode is useful for checking an NC program block by block.

Press the operating mode key.

The monitor displays the operating mode menu.

Press the SINGLE BLOCK softkey.

SELECT. PROGRAM	SELECT. BLOCK	SELECT. DELETION LEVEL
PRESE-LECTION RATE	SELECT. NC-WORD	SPINDLE SELECT.
CYCLE TIME		AUTO-MATIC-MODE

Proceed accordingly as in AUTOMATIC operating mode.

Press the AUTOMATIC MODE softkey.

SINGLE BLOCK MODE

Press CYCLE START.

Program is executed until one slide has reached the end of the block.

Press the AUTO-/ SINGLE BLOCK softkey.

AUTOMATIC

Press CYCLE START.

Program is executed continuously as in automatic mode.

Press the AUTO-/ SINGLE BLOCK softkey.

SINGLE BLOCK

Press CYCLE START.

The system switches back to single block operation and the program is executed one block at a time.

Diagnosis in side-line operation
If the diagnosis is to be selected as side-line operating mode when in the automatic mode, then proceed as follows:

Press the "continue" key in the automatic mode menu.

The follow-up menu is displayed.

Press the DIAGNOSTIC MODE softkey.

The second main menu of the diagnostic mode is displayed (see also section 7.1 DIAGNOSOS operating mode in side-line operation).

Tool inspection

With the **EPL2** control system it is possible to interrupt the program procedure in the **automatic mode** and to move away from the workpiece in order, for example, to measure the tool or to insert a new cutting tool.
Afterwards the restart program can be used to move again to the point of interruption in order to continue the program procedure.

Note: The inspection cycle can be executed **exclusively** with the tool that is momentarily in action in the automatic mode.

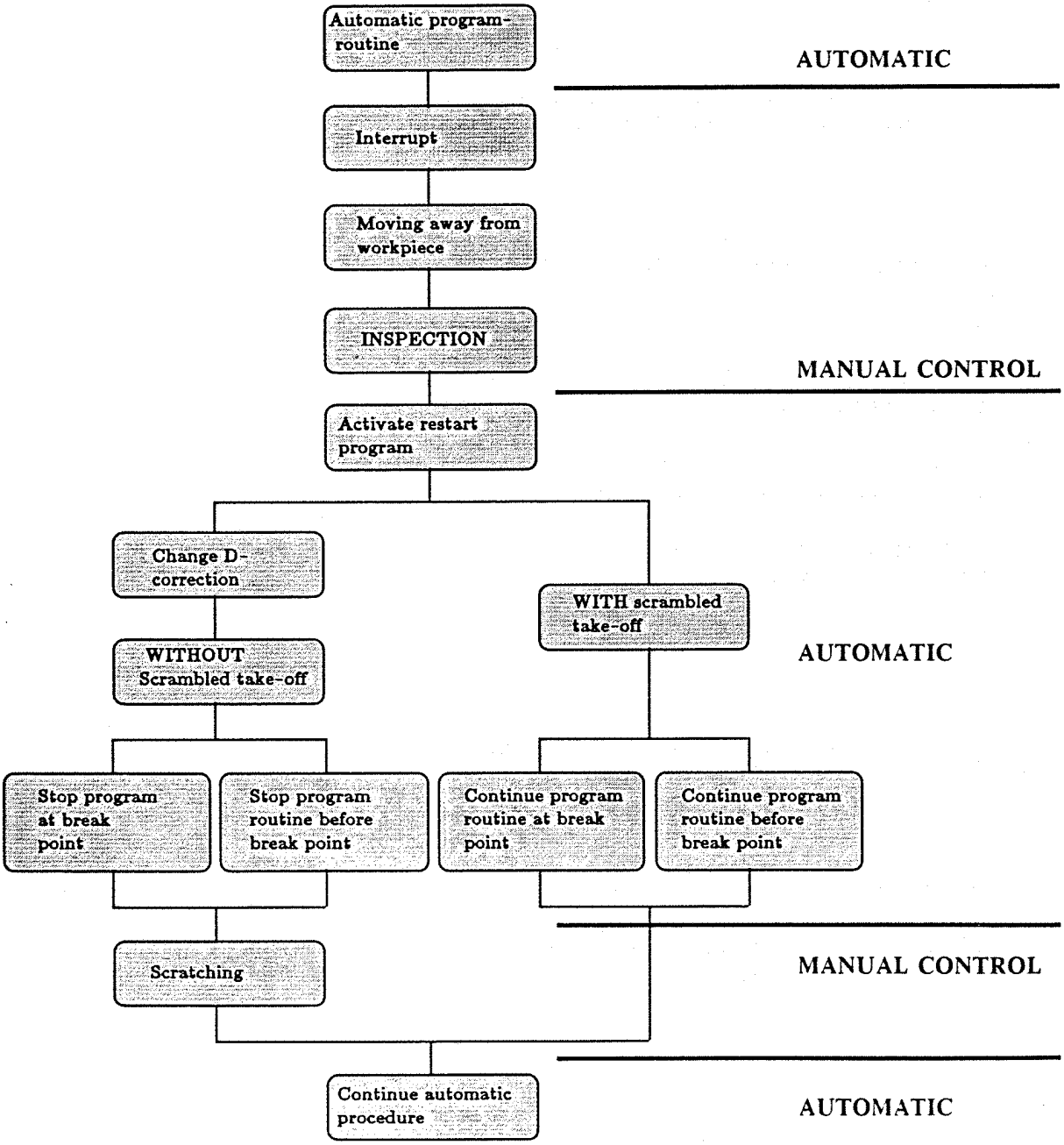
The following conditions must be fulfilled when the tool inspection is to be executed:

- The restart program **SERVICE 1** must be existent in the parts program memory.
- Actuating the operating mode key during tool inspection will **always abort** the inspection operation.
- The inspection procedure described in the following **must** always be observed.

During tool inspection the following operating functions are not allowed in the **MANUAL CONTROL** operating mode. The control system rejects them and displays an error message:

- continuous feed
- all functions of the tool setting-up mode, with the exception of the SERVICE LIFE function.

The following sketch shows the procedure and the various possibilities of the tool inspection.



Phase I (Interrupting the program and moving away from workpiece)

A program procedure in automatic mode is to be interrupted.
Proceed as follows:

Press the feedrate on/off key.

Program procedure is interrupted.

TOOL INSPECTION-TION softkey.

Change to MANUAL CONTROL operating mode

Important: If the spindle is activated, a new rotational feed can be programmed for the MANUAL CONTROL mode since otherwise the feed that was last used in automatic mode will be active.

Press the FEEDRATE MM/REV. softkey.

G95 FEEDRATE MM (INCH)/REV. F:

Enter digits.

Confirm.

New feedrate value is transferred to the control system.

Now the slide can be moved away from the workpiece, using the manual direction keys. The first five traverse paths (only traverse paths with changes in direction are counted) are stored by the control system for the restart movement.

Actuate manual direction keys.

Slide moves away from workpiece. (Traverse paths are stored by the control system).

Instructions for C-axis machining
If the function Tool Inspection is activated while the C-axis is swivelled in, the current actual value of the C-axis is stored in the variable V838 reserved for this purpose. The operator has the option of moving the C-axis in the chucking device set-up mode during the MANUAL CONTROL operating mode via handwheel or jog control mode.

Under some circumstances this function permits a convenient measurement of a milled keyway which is otherwise difficult to access, etc. when the workpiece is clamped in place. After the inspection process is over, the return program Service 1 is first used to move the C-axis to the target coordinate which was active before the interruption by entering the path command "G100 C{V838}".

Afterwards the X-axis and Z-axis are moved to the interruption point.

Important
It is not permissible to interrupt a milling operation with active milling cutter radius compensation and to subsequently re-approach with changed cutter radius, since the control must perform a completely new contour calculation in this process.

In general, we advise against interrupting a milling operation, since contour recessing always results due to the undercutting of the cutter when it travels back in.

Interrupting the C-axis machining is only practical during machining when the cutter is outside the workpiece and the C-axis is therefore momentarily being used purely as a positionin

Phase II (process of inspection)

The actual inspection process is executed (e.g. measuring the tool, change cutting plate, etc.) Tool change T1 T3 T1

Phase III (restart process)

Press the continue key.

The first follow-up menu in the main level of the MANUAL CONTROL operating mode menu is displayed.

Press the continue key again.

The first follow-up menu in the main level of the MANUAL CONTROL operating mode menu is displayed.

Press the FINISH TOOL INSPECT. softkey.

The control system automatically makes the restart program (SERVICE 1) available.
The note
****** RESTART INSPECTION ******
is displayed in the program.

After this the following possibilities exist:

1) Automatic restart with scrambled take-off

If operation in the automatic mode is to be continued with the same tool and the same cutting tool nose, then automatic scrambled take-off can be executed.

Proceed as follows:

Press the cycle start key.

RESTART WITH SCRAMBLED TAKE OFF: INPUT = 1 0.000

Enter digits.
(1)

Confirm.

RESTART BEFORE BREAKPOINT
INPUT = 1 0.000

Confirm

Before executing the stored return movement paths the machine is restored to the state (e.g.: D-values, T-values) it it was in before the interruption. Then the program procedure of the interrupted programm is continued from the point onward where it was interrupted.

If you want the program to start **before the break point**, then proceed as follows:

Enter digits.
(1)

Confirm.

DELTA WAY BEFORE BREAK POINT
IN MM: 0.000

Enter digits.

Here the distance (**Delta way**) from the breakpoint on the programmed contour is indicated at which the program is to start.

Note: maximum delta way is the distance from break point to the start of the block in which the program was interrupted.
With larger inputs restart is at the **start of the block.**

Confirm.

Restart program is executed, then the program procedure of the interrupted automatic program is automatically continued whereby the **programmed delta way before break point is considered.**

2) Without automatic scrambled take-off

If, for example, the cutting plate of the tool was changed so that tool dimensions have changed as well, then the tool concerned must be scratched again.

To do this, proceed as follows:

Before starting the restart program (CYCLE START) in the automatic mode, the corresponding D-offset must be changed in a way that the tool is stopped **in front of** the workpiece.

 Press the D TOOL OFFSETS softkey.	INPUT TOOL OFFSET NUMBER:
 Enter digits. e.g. 1	
 Confirm.	D1 DX 0.000 DZ 0.000 INPUT DELTA-DX:
 Enter digits.	
 Confirm.	DX-value is transferred and compensated. D1 DX 0.000 DZ 0.000 INPUT DELTA-DZ:
 Enter digits.	
 Confirm.	DZ-value is transferred and compensated.

then

 Press the cycle start key.	RESTART WITH SCRAMBLED TAKE-OFF: INPUT = 1 0.000
 Confirm.	RESTART BEFORE BREAKPOINT: INPUT = 1 0.000

Confirm.

Restart program (SERVICE 1) is executed. Due to the changed D-values the tool is stopped in front of the workpiece. (**After scratching,** the program procdure of the interrupted automatic program is continued from the point where it was interrupted; see next page).

If the program is to be started **before** the break point, proceed as follows:

Enter digits.
(1)

Confirm.

DELTA WAY BEFORE BREAKPOINT
IN MM: 0.000

Enter digits.

Here the distance (**Delta way**) from the break point on the programmed contour is indicated at which the program is to start.

Note: maximum delta way is the distance from the break point to the start of the block in which the program was interrupted.
With **larger** inputs the program is started at the **start of the block.**

Confirm.

The restart program is executed, **after scratching** the program procedure of the automatic program is continued, taking into account the programmed delta way before the break point.

After the restart program has finished, the **scratching process** must be selected for the MANUAL CONTROL operating mode. Proceed with the following sequence of key operations. If D is not changed, the program procedure of the interrupted automatic program can be continued directly by pressing **CYCLE START.**

TOOL INSPECTION
softkey.

Change to the main menu of the MANUAL CONTROL operating mode

Presss the continue key.

Switch-over to the follow-up menu.

Press the HANDWHEEL ACTIVE D-CORR.

The menu "handwheel" is displayed on the monitor.

e.g.:

HANDWHEEL
X-AXIS
0.01

D1 DX 0.000 DZ 0.000

Moving the handwheel.

The traversed path is displayed in the input line under DX.

HANDWHEEL
Z-AXIS
0.01

D1 DX 0.000 DZ 0.000

Moving the handwheel.

Scratching is done, using the handwheel. The traversed path is displayed in the input line under DZ.

Press the HANDWHEEL TERM. CALCUL. softkey.

The traversed path is compensated automatically in the D-offset and the actual value is corrected accordingly.

Press the continue key.

The follow-up menu will be displayed.

Press the FINISH TOOL INSPECTION softkey.

Change to AUTOMATIC mode.

Press the "cycle start" key.

The program procedure of the interrupted automatic program is continued.