

NCT®201

Controls for Milling Machines and Machining Centers

Operator's Manual

Alpha version of SW as of 24/09/2012

**Prepared based on the Operation Manual in Hungarian language, issued on 19.04.2013
and the “NCT 201_EN_NEW (7) LATHE” English Operation Manual, issued on
31.01.2013.**

**READ THIS MANUAL BEFORE USING THIS PRODUCT.
FAILURE TO FOLLOW THE INSTRUCTIONS AND SAFETY
PRECAUTIONS IN THIS MANUAL CAN RESULT IN SERIOUS INJURY**

All operators and service personnel must read this manual before operating the NCT CNC control equipment and all connected machine tools.
Keep this manual in a safe location for future reference.

Information provided by NCT relating to wiring, installation, and operation of CNC components is intended as only a guide, and in all cases a qualified technician and all applicable local codes and laws must be consulted. NCT makes no claims about the completeness or accuracy of the information provided, as it may apply to an infinite number of field conditions.

As CNC control products from NCT can be installed on a wide variety of machine tools NOT sold or supported by NCT, **you MUST consult and follow all safety instructions provided by your machine tool manufacturer regarding the safe operation of your machine and unique application.**

© Copyright **NCT** 2013.05.22

The Publisher reserves all rights for the contents of this manual. No reprinting, even in extracts, is permissible unless our written consent is obtained. The text of this manual has been compiled and checked with utmost care, yet we assume no liability for possible errors or spurious data and for consequential losses or damages. If you cannot find an evident answer to your questions in this manual, we kindly ask you to contact our experts, so that we can assist you.

CONTENTS

1. Introduction	6
2. General Operating Information.....	7
2.1. Softkey	7
2.2. Side Bar.....	8
2.2.1. WIN-S Button	9
2.2.2. Keypad	10
2.2.2.1. Buttons of the screen keyboard	11
2.2.3. Software Machine Operation Panel.....	13
2.2.3.1. Operating Modes	13
2.2.3.2. Increment Selectors	13
2.2.3.3. Feed Rate Override Switch.	13
2.2.3.4. Spindle Speed Override.....	14
2.2.3.5. Program Execution Buttons	14
2.2.3.6. Start, stop and movement buttons	14
2.3. User Box.....	16
2.3.1. Creating a New User Box.....	16
2.3.2. Setup of Password	17
2.3.3. Modifying Your Password	17
2.3.4. Cancelling Your Password	17
2.3.5. Log out from the User Box	17
2.3.6. Authorization Levels	17
3. Operation Modes	19
3.1. Jog Operation Mode	19
3.2. Incremental Jog Operation Mode	20
3.3. Handwheel Operation Mode	21
3.3.1. Multiple Hand wheels	22
3.4. Reference Point Return Operation Mode	22
3.5. Automatic Operation Mode.....	23
3.6. Manual Data Input (MDI)	23
3.6.1. Differences between Automatic and MDI Mode	23
3.7. Edit Operation Mode	24
3.8. Turn Off the Control and Restart	24
4. Position Display	26
4.1. Handling the Position Window	29
5. Status	30
5.1. Program Blocks during Automatic Execution	30
5.1.1. Program List Functions	31
5.2. Graphic Display of the Tool Path.....	33
5.2.1. Setting of the graphic display	33
5.2.1.1. Zoom	33
5.2.1.2. Moving	35
5.2.1.3. Rotation	35
5.2.1.4. Setting of the drawing	35
5.3. Macro Variables	37
5.3.1. Local macro variables #1- #33	37
5.3.2. Global macro variables #100- #499 and #500- #999	37
5.3.3. Editing of the Macro Table	38
5.3.4. Validation of Changes	39

5.3.5. The important Variables in one place	40
5.4. Messages	41
5.4.1. Construction of message numbers.....	42
5.4.2. Groups of Messages	42
5.5. Backup Copy	44
5.6. Logbook	45
5.6.1. Filtering the Messages.....	46
5.7. Parameters	47
5.8. Status Display.....	48
5.9. Part and Time Counter	50
6. Program.....	51
6.1. Single block.....	51
6.2. CNC Program	52
6.2.1. Creating a new CNC program	52
6.2.2. Editing an existing CNC program.....	53
6.2.3. Helping Functions by Editing	54
6.2.4. Program Execution in Automatic operation mode	54
7. Offsets.....	56
7.1. 0-points (Work shift).....	56
7.2. Work Offset.....	56
7.2.1. Work Offset Measure	57
7.2.2. Editing the Work Offset Table	62
7.2.3. Selection of work zero point in a program	63
7.3. Tool Offsets.....	64
7.3.1. Tool offset measuring (inside the milling machine)	64
7.3.2. Editing the Tool Offset Table.....	69
7.4. Orientation and Offset of the main spindle	71
7.4.1. Setting the orientation position	71
7.4.2. Driving Spindles in Shifted Phase Mode (part transfer between spindles).....	72
7.4.2.1. Calculation of Phase Offset	72
8. Tool management	74
8.1. Adding a new tool to the store	74
8.2. Tool life management:	80
8.3. Oversized tool	83
8.4. Deletion of a tool from the store	88
9. Program Execution Start and Stop	90
9.1. Starting the Program Execution	90
9.2. Cycle Stop	91
9.3. Reset	91
9.4. Programmed Stop M00	92
9.5. Optional Stop M01	92
9.6. End of Program: M02, M30	92
10. Program Execution Intervention	93
10.1. Conditional Block Skip	93
10.2. Increasing the Feed rate Using Rapid Traverse Jog Button	93
11. Debugging the Part Program	94
11.1. Single Block Execution	94
11.2. Dry Run (all feed rates at high speed)	94
11.3. Machine Lock Function	94
11.4. Function Lock	95

11.5. Axis Lock	95
11.6. Test Run	95
11.7. Calculation of Cutting Speed	96
11.7.1. Constant cutting speed calculation (G96, G97).....	96
11.8. Description of G and M codes.....	98
12. Interrupting and Restarting the Automatic Operation	99
12.1. Interrupting the Automatic Operation	99
12.2. Restarting the Automatic Operation.....	99
12.3. Unconditional Restart the Automatic Operation	100
12.3.1. Interrupting Simple Movements.....	100
12.3.2. Interrupting the Drilling Cycles	101
12.3.3. Interrupting the Machining in Tool Radius Compensation Mode	103
12.4. Automatic Operation Restart with BLOCK RESTART Condition	105
12.4.1. Return to the start position of the block with manual operation	105
12.4.2. Return to the start position in Automatic Mode	105
12.4.3. Different return moves while the BLOCK RESTART is active.....	106
12.5. Automatic Mode Restart with BLOCK RETURN turned on	109
12.5.1. Returning to the interruption point by manual movement	109
12.5.2. Returning to the interruption point in automatic mode	110
12.5.3. Cases of return under the BLOCK BACK condition	110
12.6. Automatic Operation Start after Block Search.....	114
12.6.1. Pointing at the Desired Block.....	114
12.6.2. Pointing at the Desired Block in a sub-program	115
12.6.3. The SEARCH Command	116
12.6.4. The GOTO Command.....	121
12.6.5. Giving a Repetition Number	121
13. Settings	122
13.1. Program settings	122
13.1.1. Style adjustment	122
13.1.2. The Sound of Error Messages	123
13.1.3. The Language of the USB keyboard	124
13.1.4. Which keyboard is active?	125
13.1.5. Freezing the windows.....	126
13.1.6. Setting the Help Text.....	127
13.1.7. Calibration of the Touch screen	128
13.1.8. Setting the time and date	130
14. Channels.....	131
14.1. Handling of the Windows	131
14.2. Position display of channels	131
14.3. The START button.....	132
14.4. STOP button.....	132
14.5. Multi-channel Program Execution in Automatic Operation Mode	132
14.6. Operation Mode Change	132

1. Introduction

Dear User,

Thank you for having chosen one of our control systems. We sincerely hope that you will be satisfied in your work with its facilities. Please remember that the skill of operating the machine can only be learned through knowing its part programming fundamentals. Similarly, no programming is possible unless the skills of machine operation are acquired.

**THE MACHINE CANNOT BE MANIPULATED OR OPERATED
SAFELY UNLESS YOU CAN INTERPRET THE PART PROGRAM
AND CHECK IT FOR CORRECTNESS! THE OPERATOR OF THE
MACHINE MUST ALWAYS FORESEE ALL POSSIBLE
CONSEQUENCES OF THEIR ACTIONS!**

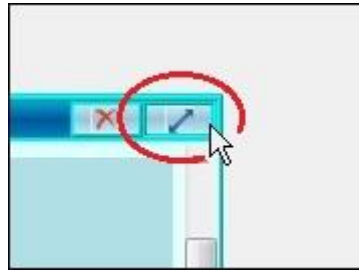
Conditions of Operation and Storage

The control system may be operated at an ambient temperature between +10°C (14°F) and +40°C. (104°F) In case the environment temperature increases above +40°C, the control must be turned off. The temperature range for storage is –10°C to +60°C. (140°F) in the electric cabinet, a slight over-pressure must be secured by means of a fan with filter, mounted on the cabinet. Cleaning and (if needed) replacement of the ventilator filter is necessary for the proper operation of the control system.

2. General Operating Information

NCT201 control units have touch screens. You can push the buttons on the screen either with your fingers or by using the stylus provided. You can use a keyboard and mouse set up via any of the four USB ports provided.

The operating surface is similar to the Windows OS. You are free to change the size of any window by clicking on the resize icon visible below.



You may close the window by clicking the red X, as shown below.



If the content inside the windowed box is not completely visible, the scroll bars on the side and bottom can be used to show all the content.

In the control unit there are two important menu systems:

- The *Side Bar*, which can be placed on either the right or left side of the screen.
- The *SOFTKEY* menu, which is displayed on the bottom of the screen.

The basic status of the **SOFTKEY** menu is shown below.

<	Status	Position	MIX	FST	View	E S C	Program	Offset	Settings	Service	Shut down	F 1
<	F1	F2	F3	F4	F5		F6	F7	F8	F9	F10	2

2.1. Softkey

The SOFTKEY menu system provides access to all of the functions of the control unit. You can operate it directly by touching the screen.

(We will use the SK abbreviation in the following text, if we describe a command line)

2.2. Side Bar

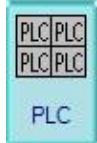
Below are all of the side bar buttons and their functions.



This allows you to move the side bar from one side to another.



This is the WIN-S button, which opens another group of windows and functions.



This is the PLC button, which allows you to switch the PLC functions on and off.



This is the software mechanical panel, which controls the machine functions.



This is the keypad button, which brings up the onscreen keyboard.



This allows you to turn the auto pop-up of the keyboard on and off.



The dock buttons allows you to place any windows on the screen next to each other without any gaps or overlapping between them.

2.2.1. WIN-S Button



The WIN-S button brings up a sub menu which contains the following 24 functions.

1. Mechanical position
2. List of G and M codes
3. Information about main spindle and tools (F, S, T)
4. Single block
5. Directory 1 (for files)
6. Directory 2 (for files)
7. Listing of activated programs
8. Graphics
9. Work zero point offset measure
10. Work zero point offsets
11. Tool correction table (mill)
12. Tool correction table (lathe)
13. Tool handling table
14. Local macros #1-#33
15. Global macros #100-#499
16. Global macros #500-#999
17. Part and time counter
18. Running applications
19. Calculator
20. Keyboard
21. Revolutions of the spindle
22. Calculation of cutting speed
23. DXF converter
24. Setup



2.2.2. Keypad

By pushing the side bar KEYPAD button, an onscreen keypad will be displayed. It is a general keypad that has numerical buttons, and lower and upper case letters according to the language selected for the control unit. It also has function keys, basic mathematical operators (+-*/) and national characters (for accents). You can find a more detailed description of the screen keyboard in the next chapter.

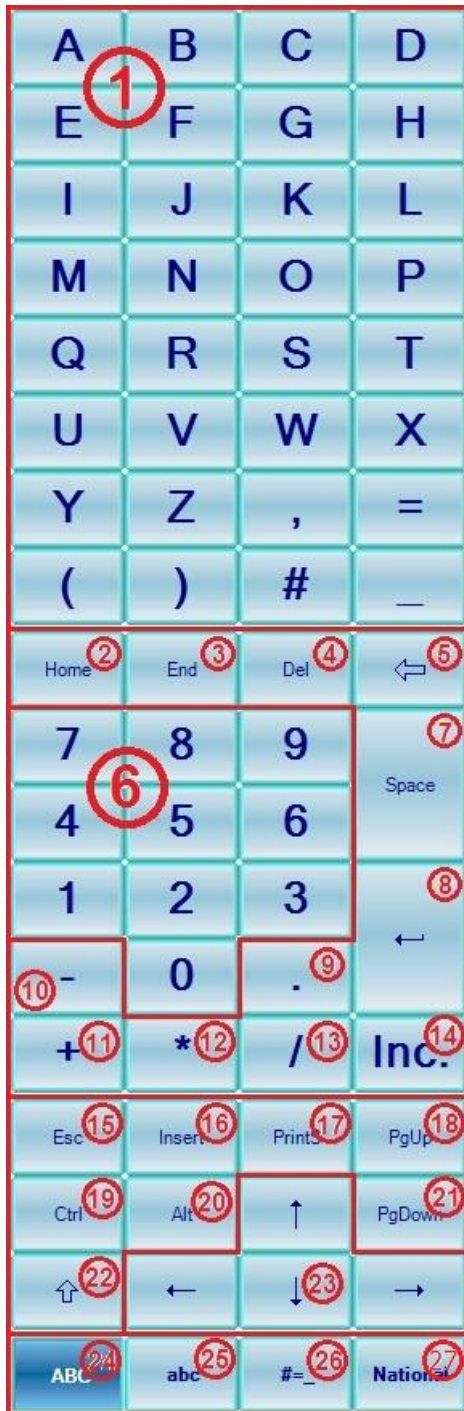
You can display another version of the keyboard in a horizontal position by using the SOFTKEY panel (basics - setup -"Soft input panel" button).



You can also connect another keyboard to any of the four USB ports.



2.2.2.1. Buttons of the screen keyboard



1) Alphanumeric keyboard: In the picture, as a default setting, in the area marked as “1” we can see the capital letters of the English alphabet. By pushing button “25” lower case letters appear. After this, by pushing button “24” we can switch back to capital letters.

2) Home: It places the cursor to the beginning of the row.

3) End: It places the cursor to the end of the row.

4) Del: It deletes the selected text or the text after the cursor.

5) Backspace: It deletes the character or the selected text before the cursor.

6) Numeric keyboard

7) Space: It puts a space into the text.

8) Enter

9) Semicolon

10) Negative sign

11) Positive sign

12) Asterisk

13) Slash

14) It serves for the entering of an incremental sign and is equal to the capital I letter.

15) ESC: It deletes the current task.

16) Insert: Switch-on or switch-off of the insertion mode. If the insertion mode is on, the program will insert the entered text at the place of the cursor. If the insertion mode is off, the entered text will replace the already existing characters.

17) Print Screen: It records the picture of the full screen and saves it into the Storage Card\Config folder in “screen.bmp” format.

18) Page Up: Scrolling up with one screen.

19) Ctrl: It is the control key, used for key combinations. For example, by the combination of keys Ctrl+C we can copy a selected text part to the clipboard. If we push it, it gets stuck, therefore we do not have to keep it continuously pushed, and upon its repeated pushing it switches off.

20) Alt: It is the control key, used for key combinations. For example, the key combination ALT+PRINT SCREEN prepares a picture only on the active window and not the full screen.

21) Page Down: Scrolling down with one screen.

22) Shift: If we push it, the Shift key will get stuck, and at the pushing of the next letter we will write a capital letter, after which the Shift key will switch off automatically.

23) Navigation keys: You can navigate the cursor by these.

- 24) ABC: In the switched on state of the button we will find capital letters and the most frequently used punctuation marks on the alphanumeric keyboard (1).
- 25) abc: In the switched on state of the button we will find lower case letters and the most frequently used punctuation marks on the alphanumeric keyboard (1).

a	b	c	d
e	f	g	h
i	j	k	l
m	n	o	p
q	r	s	t
u	v	w	x
y	z	,	=
()	#	_

- 26) #=_: In the switched on state of the button we will find special punctuation marks on the alphanumeric keyboard (1).

#	=	_	\
+	-	*	/
&		\$	~
.	,	:	;
%	@	?	!
'	"	<	>
()	{	}
[]	^	~

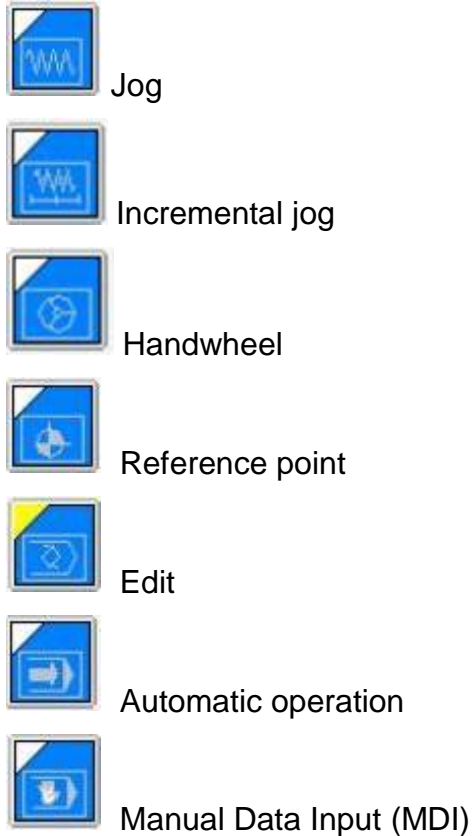
- 27) National: In the switched on state of the button we will find letters with accents on the alphanumeric keyboard (1). The button enables to write letters with accents in combination with switching on button “ABC” or “abc”. In the switched on status of the „#=_” button, the pushing of the “National” button will be ineffective.

á	é	í	ó
ö	õ	ú	ü
û			

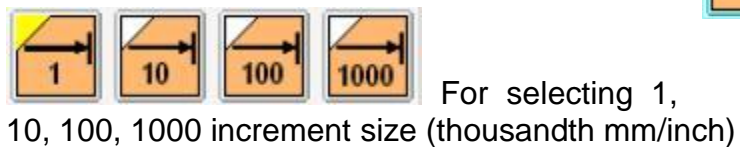
2.2.3. Software Machine Operation Panel

The software mechanical panel opens all of the functions necessary to operate the machine. The keys are exactly the same as the machine operating panel.

2.2.3.1. Operating Modes

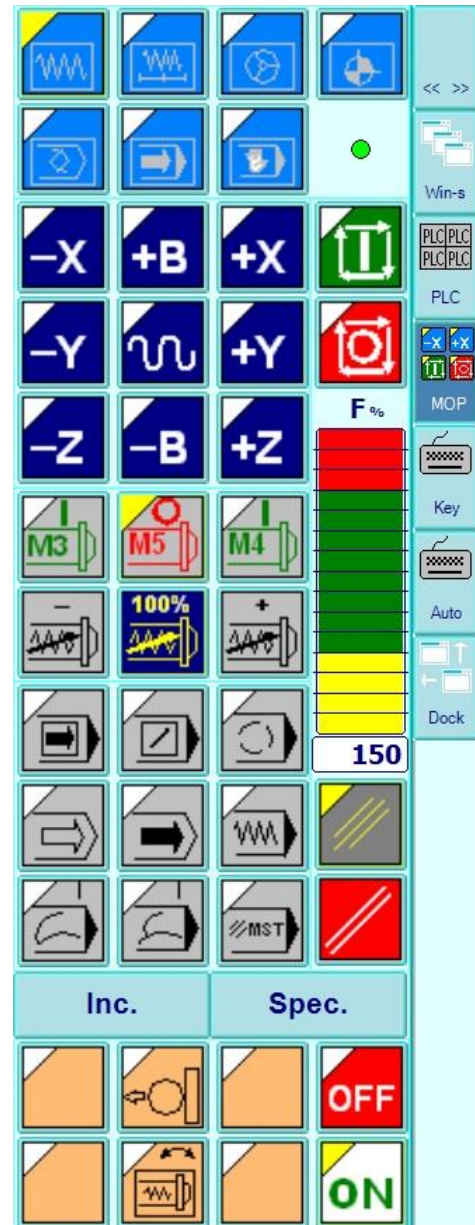
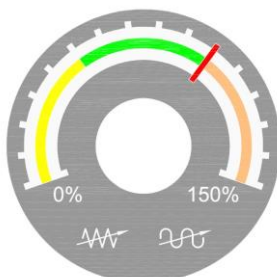


2.2.3.2. Increment Selectors



2.2.3.3. Feed Rate Override Switch

in 0 - 150% range. It can also affect the programmed feed rate of the rapid traverse.



2.2.3.4. Spindle Speed Override



The spindle RPM will be reduced or increased by 10% by pushing on the respective – or + button in the 50-150% range. By pushing the **100%** button, the programmed RPM will be set.

2.2.3.5. Program Execution Buttons



Single block execution



Conditional block skip



Optional stop



Program test



Machine lock



Dry run



Block restart



Block return



MST function lock (for testing)

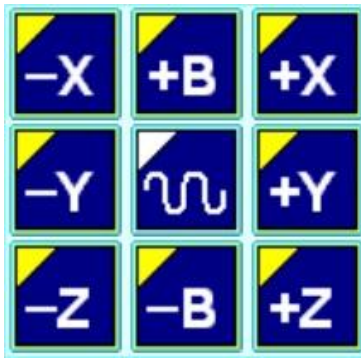
2.2.3.6. Start, stop and movement buttons



Start button.



Stop button.



Jog Buttons

The Jog buttons are used for moving the axis in a selected direction.

In case the axis moves towards a reference point, the buttons can select the axis. The arrangement of buttons can vary from one machine type to another.



Spindle Start / Stop Buttons

M3 causes the spindle to spin in a CW direction.

M4 causes the spindle to spin in a CCW direction.

M5 causes the spindle to stop.

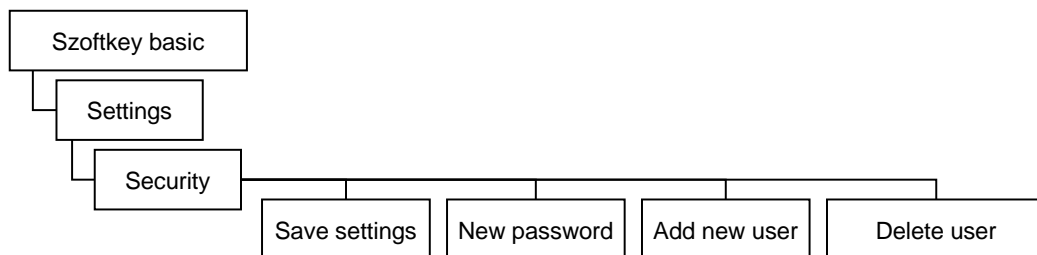
There are an additional **8 buttons with LEDs** located on the bottom of the machine operator panel that the machine tool builder can program to their choosing, with 4 of them containing optional rapid traverse override buttons among them.

A **manual handle** can be also attached to the machine control panel.

2.3. User Box

A user box is a profile set up for each individual machine operator. Each user box allows for modifications of the control unit as well as allowing additional personal adjustments (*ex.* the color of the screen, position of windows, etc.). This allows a machine to be used by several different people. Every user has access to his own user box by creating a personal username with an optional password. All users have an authorization number from 0 to 5, with 0 granting full authority and 5 granting the lowest authority. Any user may create a new user box, but he can only setup an additional user box with an equal or lesser authorization number.

Menu map:



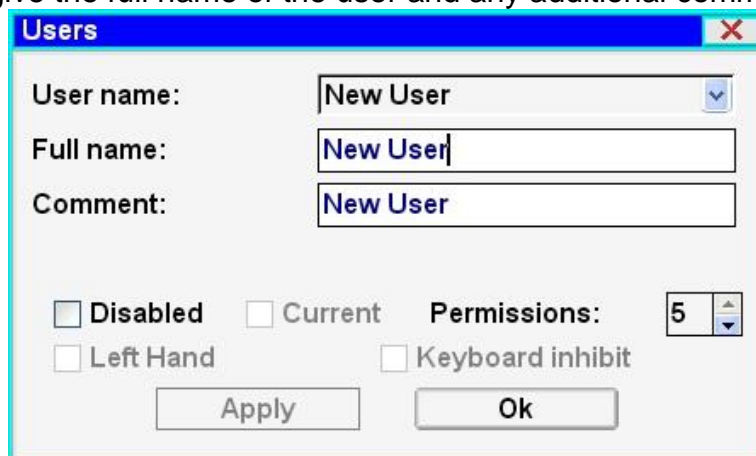
2.3.1. Creating a New User Box

To create a new box, login to an existing box and select the following options on the **SOFTKEY** menu: **SK > Setting > Security > New user**

Input a new user name and then push OK. (*Example shown below*)



Now, you can give the full name of the user and any additional comments.



2.3.2. Setup of Password

You may setup an optional password for a user box. Otherwise, everybody may access your personal user box. In order to set up a password, select the following:

SK > Setting > Security

Choose the box you wish to protect by a password, than push the button “New Password,” located in the Softkey box and then enter your password. To finalize your new password, you will be asked to confirm it by typing it again.

2.3.3. Modifying Your Password

SK > Setting > Security

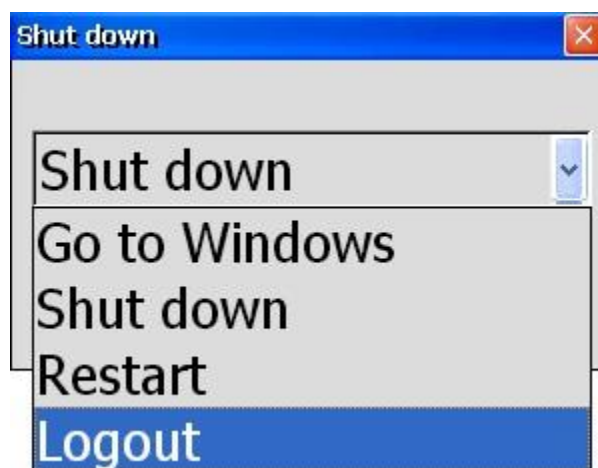
Select the user box that you wish to modify and then push the button **NEW PASSWORD**. You need to provide your old password before inputting a new one. Again, you must confirm the new password twice before it is finalized.

2.3.4. Cancelling Your Password

This is like modifying your password. Modify the old password, and don't put anything in the **New Password** field.

2.3.5. Log out from the User Box

Select the **Shut down** button from the **SOFTKEY** line. Select Shut **down** on the window, which will pop up.



Press **OK**.

2.3.6. Authorization Levels

Six different authorization levels can be used. All of them grant different rights to the user:

- **0: Full authorization** only the builder of the machine is allowed to setup a full authorization in his user box and it is only temporary. It requires a code to be

used before setting up every task. It is only valid until you restart the control unit.

- **1: Authorization** The user can set up parameters, measure a tool or a work piece, and write individual sentences. He has no right to cancel files with the “sys” extension. They can create and edit an NC program in any memory.
- **2: Authorization** The user has the right to set up parameters, to measure a tool, a work piece or to write individual sentences. He has no right to cancel file with “sys” extension, and hidden files neither. They can create and edit an NC program in any memory.
- **3: Authorization** allowing to create and edit an NC program in any memory. The user has the right to set up parameters, to measure a tool, a work piece or to write individual sentences. He has no right to cancel file with “sys” extension, hidden, archive or write protected files.
- **4: Authorization** The user can only create and edit an NC program in the directory: Storage Card / Programs. The user can set up parameters, measure a tool or a work piece, and write individual sentences. He does not have the right to cancel any files (ex. “sys” extension, hidden, archive or write protected files).
- **5: Authorization** The user can only create an NC program in the directory: Storage Card / Programs. The user has no right to set up parameters, measure tools or work pieces or to write individual sentences. He has no right to cancel a file.



Attention! There are parameter bits set in the PLC (Programmable Logic Control) that have higher authorization than the 0 level setting. If these parameter bits forbid a certain file operation, then even 0 level authorization can not modify those file operations. For more details, consult the following: **NCT 201 Parameters**

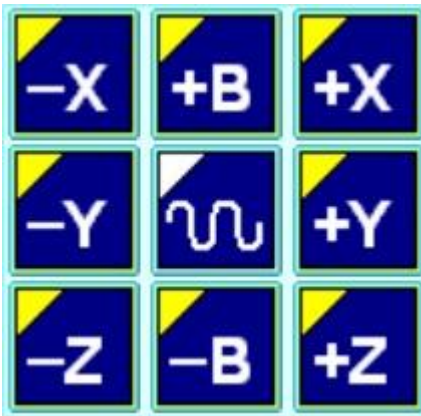
3. Operation Modes



3.1. Jog Operation Mode

Jogging can be completed only while the machine is turned on if there is no EMERGENCY STOP STATUS in the above status field. After starting the operation, **JOG** will appear in the status field.

Choice of Direction



You will find **+X**, **-X**, **+Y**, **-Y**, **+Z**, **-Z**, and **+B**, **-B** positioning buttons on the front panel. You can start movement on the **X**, **Y**, and **Z** axis directly by pushing and holding the equivalent button. You can push several positioning buttons at the same time, which will allow multiple axis movement at the same time.



Rapid Traverse Button

By pushing the **RAPID TRAVERSE** button in addition to any of the other axis movement buttons, the selected axis will move at the rapid traverse rate.

Speed Selection

The speed of the axis movement is determined on the physical machine panel or the software panel.

The speed of the **Rapid Traverse** is a fixed value that is set by the parameter axis by axis. Usually this value is smaller than the one on the **G0** rapid traverse selection.



Attention! Everything described above has been an example of the basic operation of the axis movement. The builder of the machine may create a different set of operations for axis movement on other machines using other buttons. For details, please contact the builder of the individual machine.



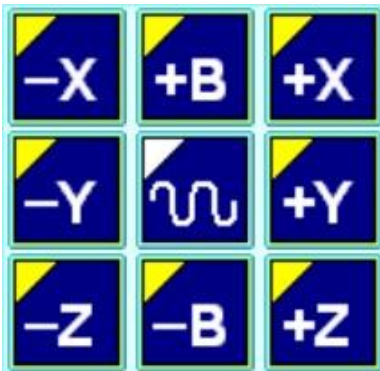
3.2. Incremental Jog Operation Mode

Incremental jogging can be completed only while the machine is turned on if there is no EMERGENCY STOP STATUS in the above status field. After starting the operation, **INCR** will appear in the status field.

In this operation mode, by pushing any direction buttons on the physical machine or software panel once, the axis will step forward according to the increment set up, with the speed of the feeding set.

If the machine has an **NCT** manual panel, the selection of the step may be done by using the stepping buttons. The size of a step made by the control unit (1, 10, 100, or 1000 increments) is always given in the least input unit of the machine. For example, if there is a metric ball screw, then the output system is metric and the least input unit definition is 0.001mm. If the control unit was set in a **G20** mode (inch) and 1000 increments were selected on the display, then the feed will be 0.0394, which is the value of 1mm given in inches.

Selecting the Direction



After setting the size of the step, then pushing the direction buttons, the stepping will be started on the selected axis and move towards the selected direction.



Note: If you use larger increments and you suddenly release the **Jog** button, the axis movement will stop and the selected distance will not be reached. In this way, you can avoid any possible collisions.



3.3. Handwheel Operation Mode

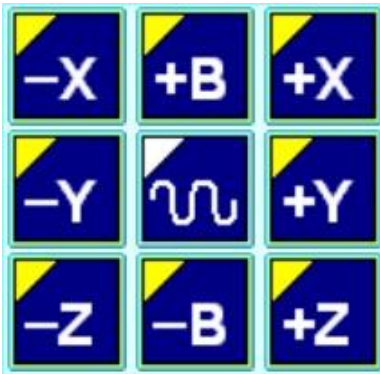
Handwheel Mode can be completed only while the machine is turned on and there is no EMERGENCY STOP STATUS in the above status field. After starting this mode, the status field will show **HNDL**.

Selecting the Step

Select the step by using the button on the physical panel of the machine or the software panel. The size of a step in the hand wheel mode shows how many increments will be stepped forward while moving the hand wheel by one pitch.

Direction

The movement of the axis is determined by the **+ / -** directions painted on the NCT hand wheel.



If the machine is equipped with an NCT machine control panel, you can also select the axis from the buttons **X**, **Y**, **Z**. By selecting the desired axis, the display for both directions (for example **+X** and **-X**) will light up.

3.3.1. Multiple Hand wheels

The control system may also be equipped with a total of 3 hand wheels. These hand wheels each control the movement of the **X, Y, Z** axes, but they are limited to each axis. If there are more axes, they have to be selected by an axis selection button and only then can they be moved using a common hand wheel.

Handwheel Movement Speed

The speed of the axes that you move with the hand wheel depends on the hand wheel step selection and how quickly you rotate the hand wheel. The axis movement starts with acceleration and stops by slowing down. There are two ways of setting the parameters for the hand wheel movement:

- The axis movement will finish according to the impulses generated by the hand wheel rotation. If the axis movement is unable to follow the impulses, then those electrical impulses will accumulate and they will move the axis after you stop moving the hand wheel.
- Even though the hand wheel generates electrical impulses that move the axis, if you suddenly stop turning the hand wheel, then the axis will also stop. Due to this, the size of the real step will not be equal to the step marked on the hand wheel.



Attention! The operation described above is simply a basic method. The final execution itself is dependant upon the design of the machine. For example, if you use a portable hand wheel instead of a built in hand wheel that is a part of the operating panel, then you would use a selector switch to determine the steps and direction of the axis instead of the machine panel.



3.4. Reference Point Return Operation Mode

To use the reference point return mode, use the following three steps:

- 1) Select the operation mode **REFERENCE POINT** by using the button on the physical panel on the machine or the software panel.
- 2) Before pushing the movement switch of any axis, please pay attention to the position of the machine and the tool. While in **REFERENCE POINT** mode, the axis will move in the direction of the reference point as long as you are pushing the movement buttons. If needed, first get to any other hand operation mode and move the tool in a position where the reference point can be reached in a straight line without crashing.
- 3) If the tool is in the proper position, then push the **CYCLE START** button, which will turn and they stay green. You will not need to move the axes one by one because they will move automatically move towards the reference point even after you have released the **JOG** buttons.



3.5. Automatic Operation Mode

A program execution in automatic mode can be completed only while the machine is turned on if there is no EMERGENCY STOP STATUS in the status field. After starting the operation, **AUTM** will appear in the status field.

The automatic mode is an operation mode for executing programs for parts. This program may run from any memory, including a flash drive or a network, but it is supposed to copy the program to the internal memory of the control unit before you can run it.

The automatic mode may be interrupted so that it can be continued later from the very same point. There will be a more precise description in a later chapter.



3.6. Manual Data Input (MDI)

This mode can be completed only while the machine is turned on if there is no EMERGENCY STOP STATUS in the status field. After starting the operation, **MDI** will appear in the status field.

In this mode, a special file called **MDIprogCH00** is executed channel by channel. The channel number is determined by adding a double digit number after **CH**. For example: **MDIprogCH00** is the editing field of the first channel for manual data input, while **MDIprogCH01** is the second channel. The content of this can be edited at your convenience according to the description you find under the chapter about editing the automatic program.

Pushing **START** in this mode will start the program, while pushing **STOP** will end it.

3.6.1. Differences between Automatic and MDI Mode

- You cannot interrupt **Manual Data Input Mode**. This means that you cannot quit and then return to continue the program from the point of interruption (contrary to automatic mode). The program will always restart from the very beginning.
- In the **MANUAL MODE** program, you cannot search by sentence.

There are certain interruptions that you can frequently use the same way, but it's necessary to only program them once. Therefore, you do not need to input individual or multiple blocks for an operation; it is enough to simply describe the necessary activity in a single program. By interrupting the **Automatic Operation Mode**, you can switch to the **Manual Data Input Mode** and start the program.

For example, if a tool gets damaged, you can start the **MDI** program, which will do the following:

1. Stop the main spindle rotation.
2. Stop the coolant.
3. Retracts the tool to a position where you can manually intervene.

You can also use the **MDI** mode to easily interrupt a part program if you need to make a measuring process. In order to do this, you need to interrupt the **AUTOMATIC MODE** and start a measuring cycle in **MDI** mode. Once the cycle is finished, you can return to the **AUTOMATIC MODE** and continue the previous program.

In the **MDI** mode, the following functions work in the same way as they do in **Automatic Operation Mode**:

Single block execution:

- **Conditional STOP**
- **Conditional BLOCK**
- **Dry Run**
- **Lock Functions**



3.7. Edit Operation Mode

This is used for program editing. The following operations can only occur in editing mode:

- Editing of the parameter memory
- Uploading or saving parameters via USB
- Uploading the **PLC** program into the control system

You can edit the programs in any operation mode, but you cannot overwrite any program that is currently in operation.

3.8. Turn Off the Control and Restart

SK > Shut down

A new window will pop up; select **Shut down** from the roll down menu, than select **OK**.



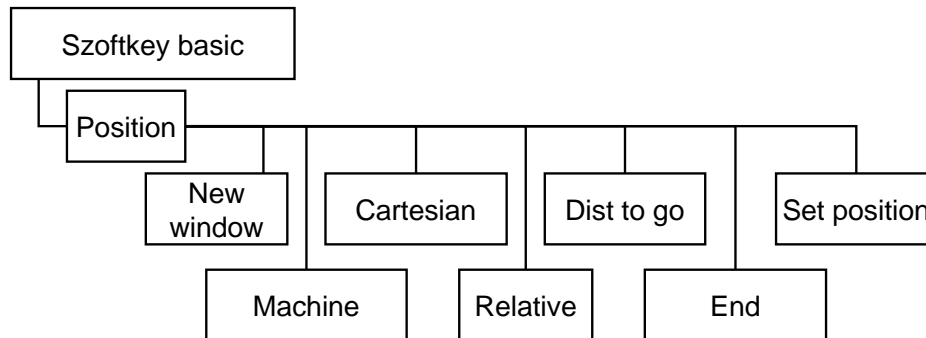
If you want to restart the control, select **Restart** from the roll down menu, than select **OK**.



4. Position Display

You can personalize the CRT monitor in such a way that all of the necessary information is always available on screen. You can collect different positions, coordinates, and other data in a single window that you can call by pushing the **POSITION** button in the **SOFTKEY** field.

Menu map:



You can combine six different kinds of axis positions while setting up a window:

Absolute Position:

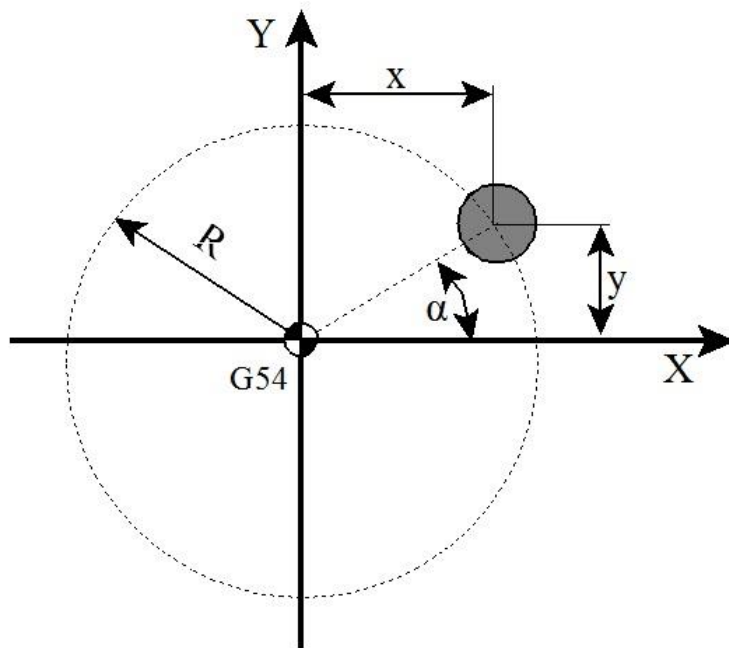
Shows the current position of the machine in the coordinate system of the valid zero point. The default code is **G54**.

Machine Position:

Shows the actual position in the basic machine measuring system. The zero points are in the reference points.

Cartesian Position:

If we use the polar coordinate interpolation, we do not write the NC program in the coordinate system (X-Y) of the machine but in a polar coordinate system ($R - \alpha$), the radius of which is R , its angle is α , and its origo is the same with the origo of the currently active zero-point. If the X-Y plain is selected, the direction of the zero angle, according to the default setting, will fall towards axis X, and the positive angle movement - viewed from the positive direction of axis Z - will rotate counterclockwise. In such a case, the commands issued for axis X refer to the radius R , and commands issued for axis Y refer to angle α . The measurement unit of R is the same with the measurement unit of X, and the measurement unit of α is degrees.



If we switched on the polar coordinate interpolation, the Absolute position will display the already programmed $R(\text{mm})$ and $\alpha(\text{degree})$ values, i.e. the position of the machine interpreted in the polar coordinate system, instead of the positions of the usual $X(\text{mm})$ and $Y(\text{mm})$ axes.

By the “Rectangular position” button we can reset the display, thus instead of the polar coordinates we will see the numeric values of the rectangular position of the machine.

Relative Position:

After reaching the reference point, the relative position is the same as the absolute position. This can be overwritten and set to zero in any preferred position. You can use it to measure the axes movement, but you cannot refer to this coordinate system with an NC code.

Distance to Go:




This shows the distance from any position to the end point of the programmed position.

End Position:

This is final point of a programmed axis movement in a block. This position is within the actual selected coordinate system of the part program. The geometric offsets of the tools are also taken into account in this position.

Length correction display:

In the Position window, on axes where a length correction is redeemed, the axis name can be seen in a frame. By touching inside of this frame, instead of the position the length correction number and value will be displayed. (The amount of the correction and wear with a sign) The length correction of all axes can be displayed at the same time by the “Length correction value” button. (Please see also the next chapter.)

Absolute position  			
Absolute position G54		Dist to go	Endposition
X+	0.000 _{mm} +	0.000 +	0.000
Y+	0.000 _{mm} +	0.000 +	0.000
 Z	H1=11.000 +	0.000 +	0.000

4.1. Handling the Position Window

The position window will be displayed by pushing the **POSITION** button in the **SOFTKEY / BASIC** menu. It can also be displayed through the **WINDOWS** menu on the sidebar (Win-s). On the header of this window, you will always see the format of the current position. You can personalize the position window in order to find the information that you need. You can select one position from the following: **ABSOLUTE**, **MACHINE**, **CARTESIAN**, and **RELATIVE**. If you need to select any more positions, you can do this by pushing the **NEW WINDOW** button.

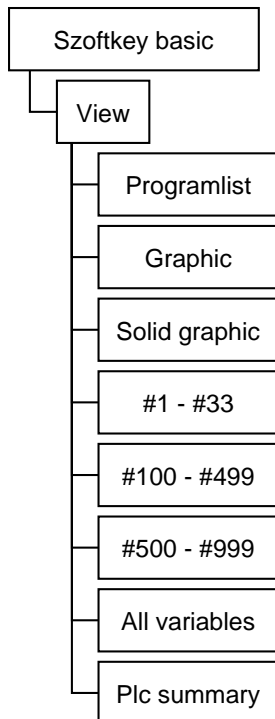
Once the **Machine Position** button has been pressed down, the display header will be highlighted and the machine position will appear. If the button is pressed again, the display header will revert to the absolute position based on the current zero point.



5. Status

The View menu brings up a new row of Softkey buttons, which displays the status of all currently running programs.

Menu map:



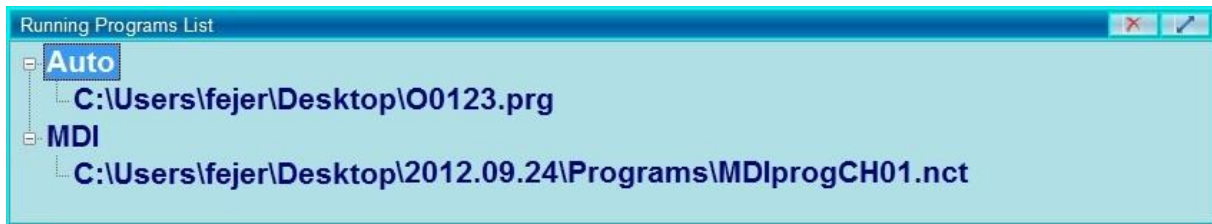
5.1. Program Blocks during Automatic Execution

To display the currently running program, push the **Program List** button, which will bring up another window (see below) that shows that current program's blocks. The block under execution is highlighted in bold. This window's appearance is determined by selecting either the **AUTOMATIC** or **MDI** mode.



If you need to see which program is running in **AUTOMATIC** or **MDI** mode, then you must select the **RUNNING PROGRAMS** button, which is located on the **SOFTKEY** menu.

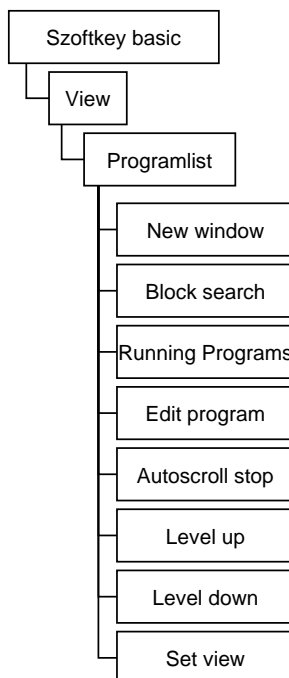
Whichever program is running, the name of this program is highlighted in the list.



5.1.1. Program List Functions

If the window **Program List** is on display, the header will get darker when it is selected and the following functions will appear in the **SOFTKEY** line: (If you want to change to another window, touch the header of the desired window)

Menu map:



New Window

By pushing this button, the old window will continue to be on display, but a new window will also appear with similar content (it could be in front of the old one). The windows can be moved next to each other by grabbing their header. This way you can see distant program lines at the same time. It can also be useful at the jump commands within a file (GOTO) because you can see the line containing the jump command and the context of the block where you have to go to.

Block Search

The Block Search will work only in **Auto Mode**.

It will not work if the following sub menu buttons are shaded:

- Cycle start on
- Cycle stop on
- INTD status (program interrupted)
- Previous block search is active

With active block search, a text field will appear on the bottom of the **Program List** window. Input the text you are looking for and if you get multiple results, you can move among them by using the **>>** button. You can also go backwards by pushing the **LAST** button. If you select the Row number soft key, the number that you typed in to the text field will lead you to that row. If there is no such block number, then the **Not Found** message will be displayed. It is also possible to search for a block by clicking on the arrow on the left side of the desired row, or by using the stylus of the NCT control touch screen.

If the searched block was a sub-program call, the search can be continued in the sub-program. The **Goto** soft key button will pop up a text field with the following question:

When you select **GOTO** it does not execute the sentence which is before of the selected line!
Are you sure you want to continue?

Your search could also be case sensitive if you use lower and/or upper case letters in your program.

Running Program

A window will pop up to show the programs running in **Automatic** and **MDI** modes.

Edit Program

The program that is in the Program list will pop up in a new window with Edit on the header. You can then edit and save the program.

There are other new buttons that will appear on the **SOFTKEY** line:

Run - You can assign this program to run.

File - The edited program can be saved with the same file name (using Save) or with a different file name and location (using Save As).

Edit - This will create another set of keys that will you: Undo, Cut, Copy, Paste, Select all, Search/Replace functions

Insert - There are several additional insertion functions that are used for faster editing.

Block Number - This allows you to renumber the blocks according your selection or it will be use auto numbering.

Word Wrap - This will allow you to format long text into a shorter column.

Autoscroll Stop

When activated, this will display only a portion of the program that is visible in the window. Otherwise, all blocks of the program are visible as the program execution continues.

Level up, Level down

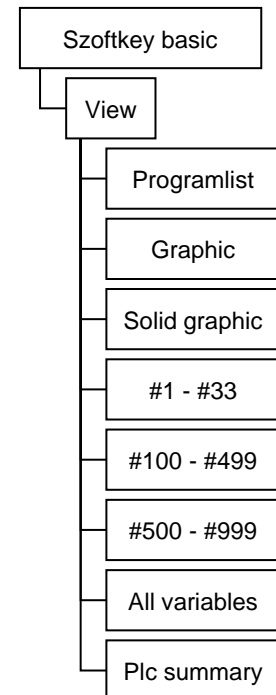
By using these buttons, you can control which level of the program is displayed. This can allow you to see the running macro program from the level of the calling or in greater details if you chose to go within the sub-program.

5.2. Graphic Display of the Tool Path

SK > View > Graphic

The absolute position can be displayed (independently of the operation mode) in the graphic window. The window will be on the lower right part of the screen. The tool path will be shown as a continuous track. The rapid traverse section will appear with a red line and you will see the normal feeding as a green line.

Menu map:



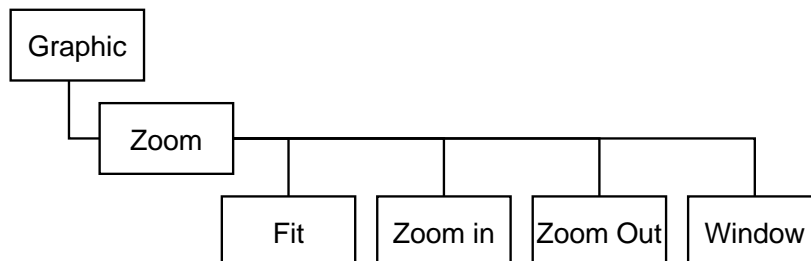
5.2.1. Setting of the graphic display

The lines will show the model of the real tool path. You can select the choice of a display.

In the menu “**View setting**” you can select several views, such as an isometric or a Cartesian view or a view of any of the selected planes XY, ZX, or YZ. You can even adjust the rotation of the coordinate system. The direction of their axes can also be determined by the following presets: XY Set, ZX Set and YZ Set.

You can undo or redo the adjustment of the view by selecting “**Undo**” and “**Redo**” buttons.

5.2.1.1. Zoom



You have several possible ways of zooming in on the display.

1) SK > View > Graphic > Zoom

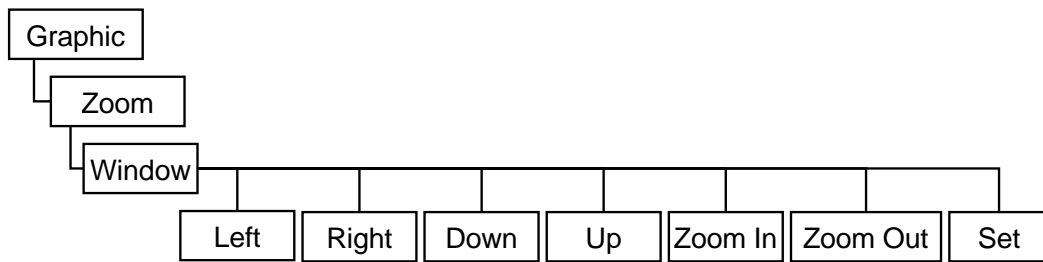
Fit - The whole tool path will fill up the size of the window.

Zoom In - The tool path will be larger, starting from the central point of the window.

Zoom Out - The tool path will be smaller, starting from the central point of the window.

2) SK > View > Graphic > Zoom > Window

A yellow rectangle will appear on the drawing. Using the buttons **Zoom In** or **Zoom Out**, you can adjust the size of the window. The “Left”, “Right”, “Up” and “Down” buttons are for adjusting the position of the window. Press the “Set” button and the content of the selected window will fill the whole screen.



3) Directly on the touch screen you can circle the area you wish to enlarge.

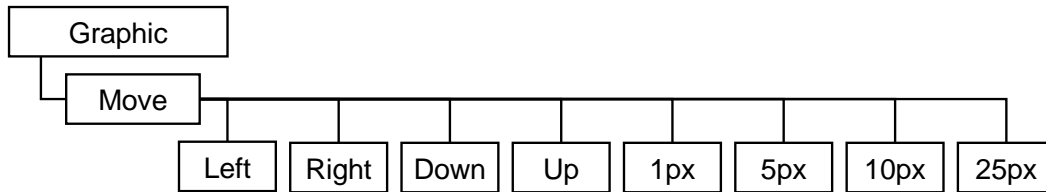


You can see a either a magnifier icon or a movement icon in the left upper corner of the Graphics window.

Touch the magnifier icon and circle the area you want to enlarge.

After releasing the screen, the content will fill up the screen automatically.

5.2.1.2. Moving



You can move the drawing two different ways in the Graphic window:

1) SK > View > Graphic > Move

Use the function of the buttons „Left“, „Right“, „Up“, „Down“ to move the drawing. The speed of the movement depends on the button turned on (1 px“, „5 px“, „10 px“, „25 px“). The one that is being used has a darker background. For example, if the button „10 px“ is on, the drawing will move left by 10 pixels when you the Left button.

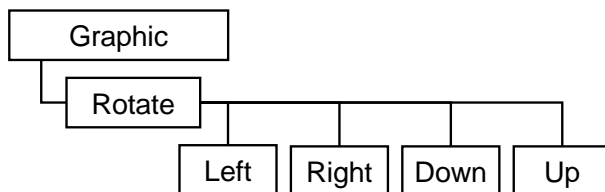
2) To move the drawing directly on the touch screen:



Select the magnifier icon. Trace a line on the touch screen in the direction that you want the drawing to move towards.

After releasing the screen, the drawing will move along the line you traced.

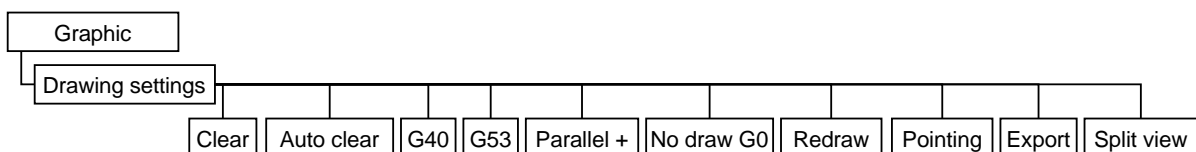
5.2.1.3. Rotation



The movement is similar to rotating a globe, in which case the center point is locked down. If we move the front section down, the rear section will move up. Likewise, if we move the front section to the right, the back section will move to the left.

5.2.1.4. Setting of the drawing

You can continue to further fine-tune the ways in which the graphics are displayed by using some of the following settings:



Clear:

Cancels the current drawing.

Auto Clear:

Cancel the drawing which was made in the previous program. A new drawing will start when the program begins.

No draw G0:

The movements made in rapid traverse won't be drawn.

Export:

You can save the content of the graphic window as a picture. Upon pushing the button, a directory will show up. Please give a name and a format to save the picture.

Redraw:

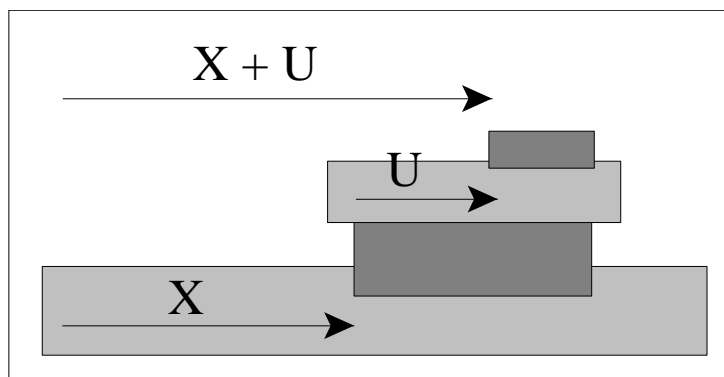
This updates the drawing by redrawing it.

Sketch:

You can draw a sketch in the graphic window.

Parallel + :

In case of parallel axes, it is fixed on a parameter which one is the basic axis from among axes parallel with each other, and which ones are the "parallel" axes parallel with the basic axis. If the parallel axes are situated in a state where they are mounted on each other, by the "Parallel +" button it is possible to have their position drawn in a summarized way in the Graphics window.

**Pointing:**

Instead of straight lines, dotted lines will be used for drawing.

Split view:

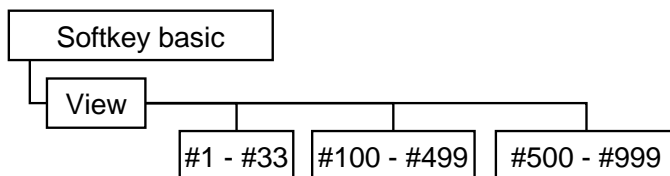
While this button is turned on (indicated by a darkened background), three typical views will be shown at the same time. Three different plane views (XZ, YZ, and XY) and a fourth view that is freely adjustable. If you turn on the "Split view" function, you will have to adjust the fourth view again because the previous adjustment will be lost since the function was turned off.

G53:

The program parts interpreted in a different zero point system will be shown in a machine G53 coordinate system. When switched off, the program will be displayed without a work shift.

5.3. Macro Variables

During programming, it is possible to use macros. You can give them a value, read their content or edit them under a table form. By assigning a number, you can give a value directly to that macro variable. In the CNC program you can also give a value to a macro variable indirectly. This means that instead of giving a value, you can give the variable that contains the value. For example, you can give only the variable #1–#33, #100–#499 or #500–#999. Local and global macro variables can be distinguished by different numbers and “0” and “empty” are also distinguished as numbers.



5.3.1. Local macro variables #1- #33

During programming, you can use sub-program calls. These sub-program calls can be nested inside 16 levels. Local macros have to be considered as separate macros on all levels so that they can represent the values of the current level. The names they have are the same at all levels. If you make a link, you reach the local variable of the level where the subprogram was called.

While starting a program, the first level of the local macro variables will be totally cancelled. Every time a new sub-program is called, the new level will be cancelled as well. The previous content will be lost and it will be loaded by the programmed values while the subprogram is running. The content of the local macro variables will also be lost at every level while turning off the control.

For example, you can see the macros #1- #33 in the table below and you can edit them.

SK > View > #1- #33.

Local macro variable #1-#33				
Variables	Level 00	Level 01	Level 02	Level 03
#1 (A A)				
#2 (B B)				
#3 (C C)				
#4 (I I1)				

5.3.2. Global macro variables #100- #499 and #500- #999

Global macros are the same variables at all of the 16 levels of the subprogram calls. If a link is made, you reach the same global variable at every level. The content of the global macro variables #500–#999 will not be lost upon turning off the control. You can see the macros for #100-#499 and #500-#999 in a table and edit them under the following chain: **SK > View > #100-#499** and: **SK > View > #500-#999** respectively.

Global macro variable #100-#499		
Variab...	Value	Comment
#100		
#101		
#102		
#103		
#104		

Global macro variable #500-#999		
Variab...	Value	Comment
#500		
#501		
#502		
#503		
#504		

5.3.3. Editing of the Macro Table

New Window

By pushing the **New Window** button, the old window will continue to be on display, but a new window will also appear with the same content (although it may hide the old one). The windows can be moved next to each other by grabbing their header, allowing the display of local variables at different levels in multiple windows.

Local macro variable #1-#33		
Variables	Level 02	Level 03
#31 (- I10)		
#32 (- J10)		
#33 (- K10)		12

Local macro variable #1-#33		
Variables	Level 00	Level 0
#1 (A A)	43	
#2 (B B)		
#3 (C C)		

Import

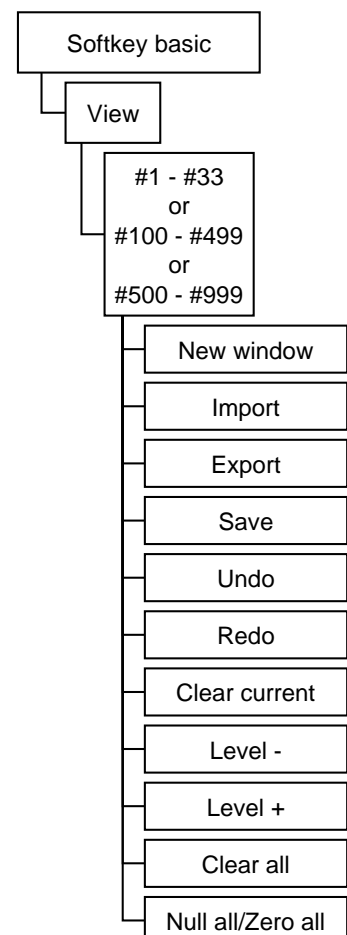
The already saved table with macro variables can be imported by using the **Import** button, which can be found in the **SOFTKEY**. After importing the variables, they will be saved.

Export

If you wish to create several different macro tables, you can export it by pushing the **Export** button, which can also be found in the **SOFTKEY**. Once pressed, a directory will appear that will allow you to name and save the macro table.

Undo

You can undo the previous modification by using the following command chain: **SK > View > #100-#499 > Undo**. A window will pop up that contains the previous modifications. Then you can search for the modification you wish to undo by using the **Up** and **Down** buttons in the **SOFTKEY**. Once you press the **Undo** button, an



alarm window will pop up with the question: "Do you really want to restore the selected variables?" Press "Yes."

Macro variables Undo						
Number	Channel	Offset index	Private	Old value	New value	Comment
1	1	#1	Level 00		43	#33 edit
2	1	#33	Level 03		12	#33 edit

Redo the Undo

In order to redo a previous undo, press **Cancel** in the **SOFTKEY**. A window containing a list of the previous redoes will pop up. Simply select the one you wish to redo by using **Up** and **Down** buttons in the **SOFTKEY** and then press **Cancel**. An alarm window will pop up with the question: "Do you really want to restore the selected variables?" Press "Yes".

Level +/-

If a local macro table is selected, you can switch from one level to another one by using the **Level + / -** buttons.

Clear Current

If you need to delete a variable on a macro table, select the variable and press **CLEAR**.

Clear All

If you need to delete and empty an entire macro table, simply press the **Clear All** button. A new window prompt will ask, "are you sure?" Press "Yes".

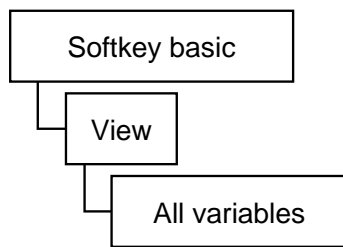
Null All/Zero All

If you need to set the entire macro table to 0, press the **Null All** button. A new window prompt will ask, "are you sure?" Press "Yes".

5.3.4. Validation of Changes

The changes you have made in a CNC program will be validated immediately, but not all at the same time. The control unit will upload macro table by the values programmed continuously while the program is running. By calling the subprogram, the new level of local macro variables will be automatically cancelled in the table. Modifications that were made by editing the table will not be necessarily validated at once. While the program is running, if you can see that the next 2 or 3 lines in the program will refer to a macro variable, then you can still change those variables in the table. The old value may still be used during the program execution because the program is not yet at that sentence, even though the control has already buffered that sentence.

5.3.5. The important Variables in one place



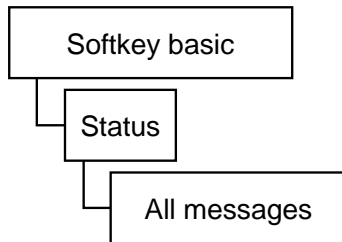
If you need to collect the most important variables in one window, use the following command chain: **SK > View > All variables**

The **All Variables** window will pop up, allowing you to follow the value of 20 macro variables at the same time. Simply enter the name of the variable in the first column you wish to check and on the right side you will be able to continuously view its current value.

5.4. Messages

If you need to see all of the control's messages, use the following command chain:

SK > Status > All messages



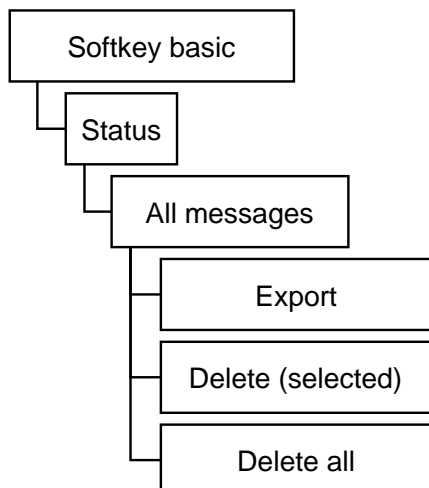
The window that pops up is called the **Global Errors** window. This same window can be called by double-clicking on the message field in the status bar.

00100201	X Axis Servo error 2	12.03.2012 01:15:23 du.	Channel1 Mill	13:15:41
EDIT		mm	EMG	

In the message field, you can see the last error message. If there are more than one error messages, you will find a number before the text of the message, displaying the number of the messages. Between two ▼ signs, you will find the text of the last message. The ▼ signs also indicate that there are further messages.

00905200	(2)▼ Last workoffsets loaded: 'WorkOffset.old' (02.08.2012 10:36:35 de	▼ 12.03.2012 01:05:54 du.		
EDIT		mm		EMG

Following the **All Messages** command chain through the **SOFTKEY** line, you can find the following functions:



Export

This allows you to save the content of the Global errors window. Simply push the **Export** button and a directory will show up. Give a name and a format in order to save the error message.

Delete (selected)

To cancel the selected error messages, press **Delete (selected)**.

Delete all

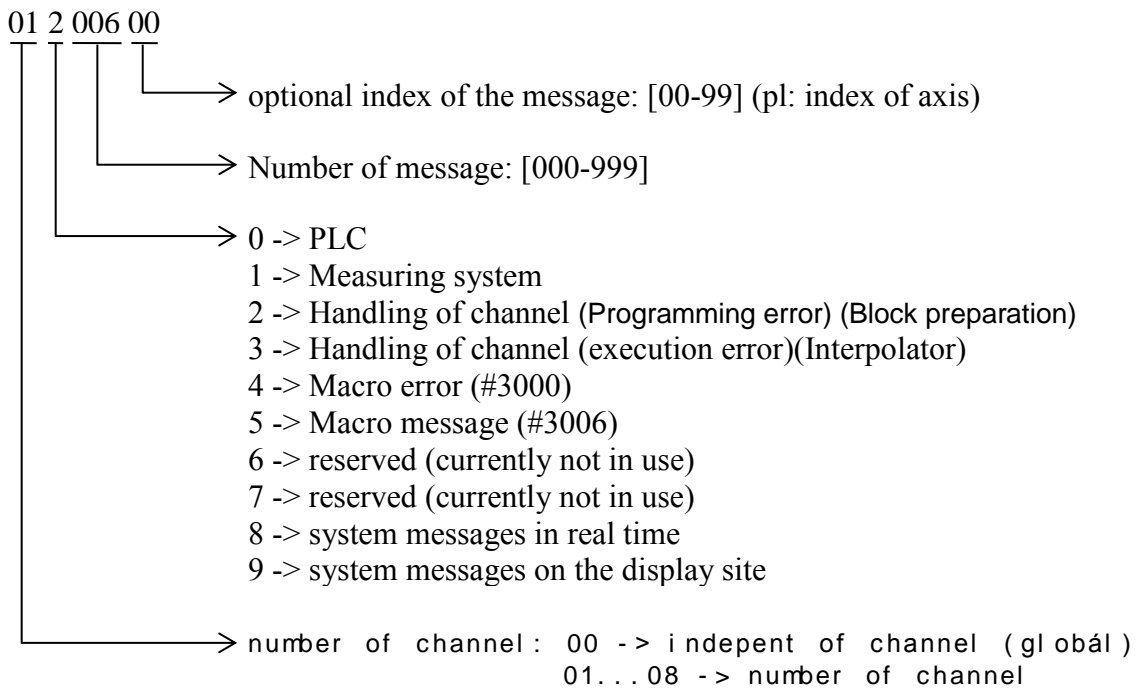
To cancel all the error messages, press **Delete all**.



By pushing the **CANCEL** button, the selected message (by default, the last one) will be cancelled. The last message appears also in the first line of the status field. By pushing **CANCEL** you can cancel the messages one by one and you won't need to call the window "All messages".

5.4.1. Construction of message numbers

Exemple: 01200600 Illegal G-code



5.4.2. Groups of Messages

Messages can be divided into the following main groups:

System Alarms:

- Alarms due to failure or false setting of servo system
- Alarms due to failure of position encoders
- HW/SW alarms coming from the NC

Other NC error messages

- Reference point return errors
- Errors due to moving to overtravel positions and forbidden areas

Messages sent by the PLC program

- There may be alarms during the operation of the machine
- Messages informing the operator

Error messages sent by program module buffer processor

- If the program module buffer processor finds an error in the pre-processed program block, an error will be generated

Error from the user macros

- Error messages which are programmed by the user and will be cancelled
- Messages waiting to start, also programmed by the user

Possible answers for the messages:

Cancelling messages by turning the power off and on

In case of extremely serious or dangerous alarms, the message cannot be cancelled and the CNC must be restarted. These messages will cut the machine off from the CNC and create an emergency stop state (**EMG**) every time.

Cancelling messages by using the CANCEL button

This is the general way to cancel an error message. In case of extremely serious or dangerous alarms, the message cuts the machine off from the CNC and creates an emergency stop state (**EMG**) and then deletes the reference point. After using the **CANCEL** button, the machine can be turned on but then it must be returned to the reference point.

Cancelling messages by means of pressing the START button

Certain messages coming from the PLC program (or those created by custom macros) will send the control to a **STOP** state to await the operator's intervention. After this operation, the machining will continue once you press the **START** button.

Cancelling messages by the operator's intervention

Certain messages can only be canceled by eliminating the reason of the error. For example, the message **LIMIT X+** can only be canceled if the slide is moved in the opposite direction manually. The limit switch can be cleared this way. All the other LIMIT type errors can also be cleared this way.

5.5. Backup Copy

To make a backup copy of any general data, use the following command chain:

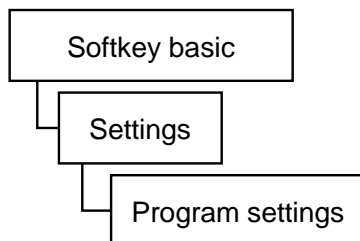
SK > Settings > Prog settings

In order to avoid doing this operation one by one, you can make a backup copy from the whole system, containing the following:

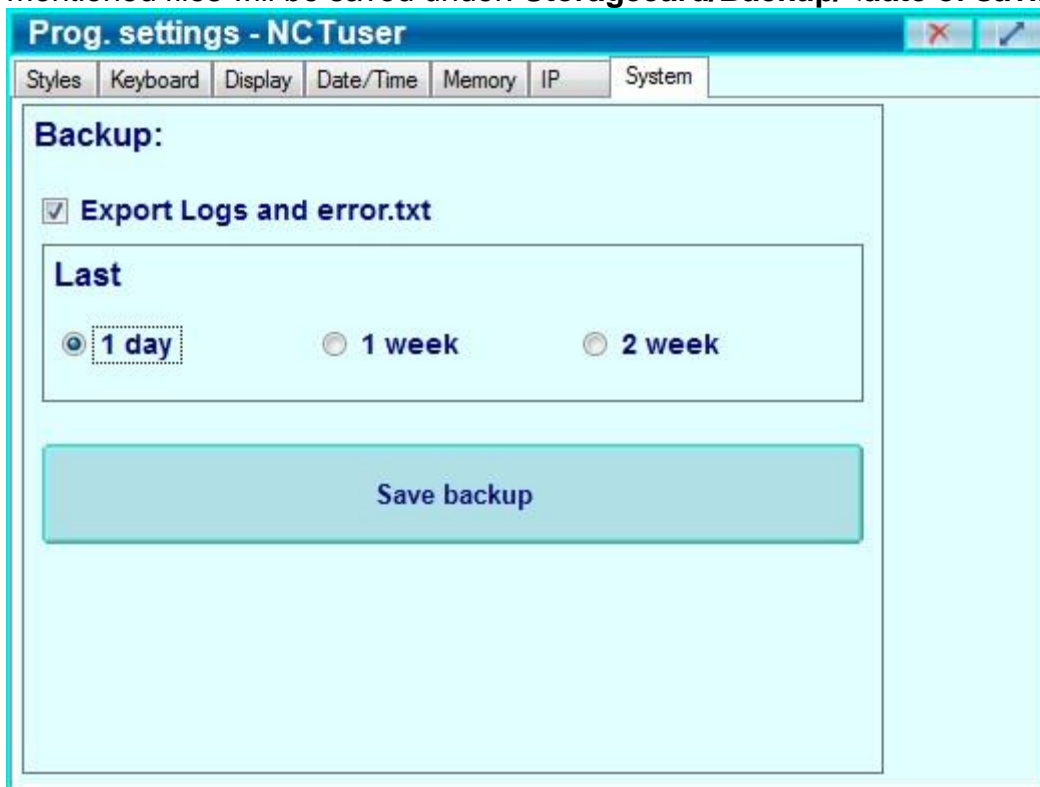
#500-#999	nct
G54-G59	nct
Mill_228_T	plc
Logs	txt
Nct200Win	reg
Param	par
Toolodb	csv
TooloffsetM	nct

- macros #500-#999
- every work shift (not just G54-G59),
- plc program,
- messages,
- Windows registry,
- parameter tables,
- tool storage tables,
- tool offset tables

Menüterkép:



In order to backup the data, first select the **System** tab in the pop up window (shown below) then select the time frame of the message that you are saving and check *Export logs and error.txt*. and then press the **Save backup** button. The above mentioned files will be saved under: **Storagecard/Backup/<date of saving>**

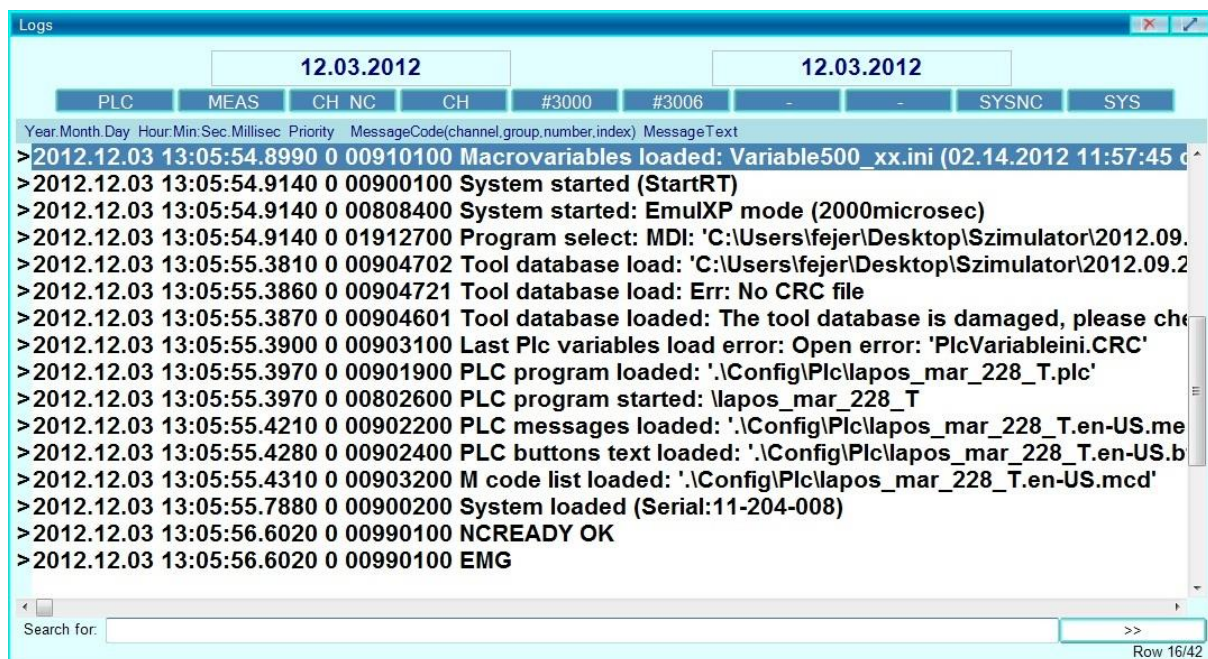
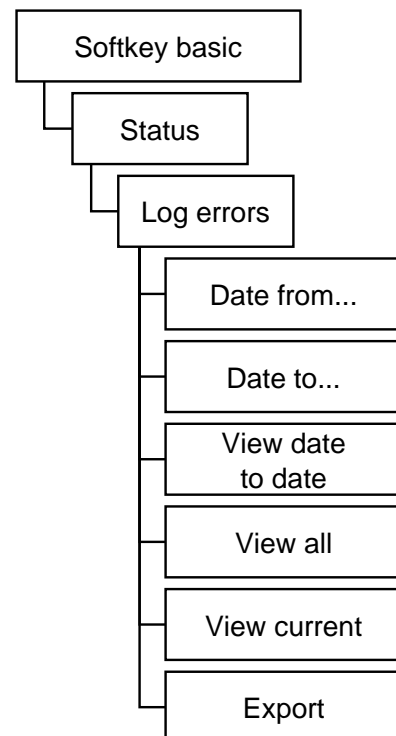


5.6. Logbook

The logbook sees and recalls everything that is happening in the control. The error logbook can be saved instantly and forwarded to the NCT service center. The error logbook file helps the repair work get finished faster and cheaper, allowing the down time of the machine to be reduced considerably.

In the logbook registers the following: changing the operation mode, when the starts and stops, the waiting periods, all alarm messages, tool changes, work shifts, etc. The old and new values of the modifications are recorded also. Various messages from the PLC and the CNC are recorded as well.

The complete operation of the machine can be reconstructed precisely and quickly. The error diagnostic is also easier to use.



5.6.1. Filtering the Messages

You can filter and relevant messages in the logbook by date or message group.

Selecting the Starting Date

Double click on the left side date button: **Starting Date**. A calendar will appear in the middle of the window.



Select the starting date.

To find the ending date, double click on the right side date button: **Ending Date**. A calendar will appear in the middle of the window, which causes the starting date window to disappear. Select the ending date.

You can also select the starting and ending date by the **Date from** and **Date to** buttons.

In order to display all of the messages between the selected date period, push the **View Date to Date** button.

In order to display all of your messages in the logbook, select the **View all** button.

In order to display all of the messages since the machine has been turned on, select the **View Current** button.

Selecting the messages by the groups

The buttons under the date periods will show the different message groups.

PLC	PLC
MEAS	Measuring system
CH NC	Handling of the channel (block preparation)
CH	Handling of the channel (Interpolator)
#3000	Macro error (#3000)
#3006	Macro message (#3006)
-	- (currently not in use)
-	- (currently not in use)
SYS NC	system messages in real time
SYS	system messages on display side

By pushing the buttons (the header will be darkened), the messages in the selected group will show up. Messages related to the off buttons won't appear in the logbook window.

Order of the message entry

The message entries will follow each other in a timely order in the log.

As a display option, you can find two buttons on the **Softkey** line that can be used to set the direction of the time

From up to down  Earliest date on the top

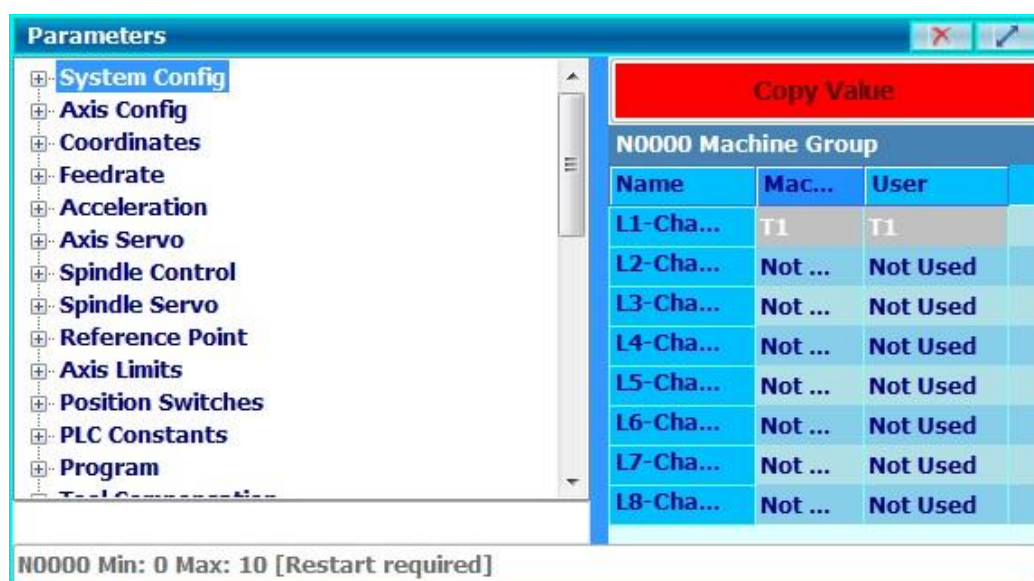
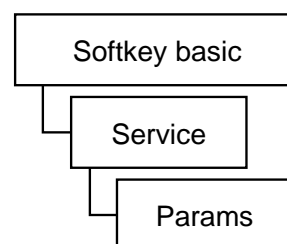
Form down to up  Latest date on top

Logbook File Preparation

Press the button **Export** to begin file preparation. In the window that pops up, give the name and place of the file, then push **Save**. Only the content of the window – according to the selected requirements – will get into the logbook file.

5.7. Parameters

The parameter functions make it possible to view, modify and save the values, adjustments and parameters of the control unit and PLC. For more details on how to use the parameters, see the diagnostic description of the control NCT2xx.



5.8. Status Display

The top line of the screen is the status window, You will get a global view of the current state of the control and the machine. This display field location is static, regardless which window you open.

In the first line you will find a message field. In this field the followings will be displayed: operating messages, errors of the CNC, and macros and PLC messages. You will also find the date and time in the same line.

00100201	X Axis Servo error 2					12.03.2012 01:15:23		Channel1 Mill	14:32:52
1	2	3	4	5	6	7	8	9	

The further lines belong to the channels. (i.e. If there is a one-channel machine, you will have only two lines). There are nine status fields and in the last field you will find the path and name of the program selected on the current channel for the automatic execution. In manual data input mode, you will find the path of the file to be used for manual data input.

Possible Status Field States

The lines determine priority, with the top line requiring the most priority and each consecutive one requiring less. If multiple statuses occur at the same time, a higher level state will overwrite a lower level state.

1. field	2. field	3. field	4. field	5. field	6. field	7. field	8. field
EDIT	SBLOCK	NSCH	INTD	(DWL) [S]	PLC	#	NC
AUTM	JOG	GOTO		STOP	DRY RUN	*	KLAV
MDI	INCR	START		F0%		®	PLC
	HNDL	STOP		F=0		!	EMG
	REF			MOV			FLOCK
	SBEX			POS			TEST
				0.001-1			MLOCK
				mm/ inch			REF

1 First status field states

- **MDI**: Manual data input mode
- **AUTM**: Automatic operation mode, program execution from the memory
- **EDIT**: Edit mode

2 Second status field states

- **JOG**: Jog
- **INCR**: Incremental jog

- **HNDL**: Handwheel
- **REF**: Reference point return
- **SBEX**: Block restart

3 Automatic operation and manual data input mode

- **START**: Start
- **STOP**: Stop
- **NSCH**: Block search
- **GOTO**: Go to a block

4 Fourth status field states

- **INTD**: Automatic execution interrupted

5 Interpolator states

- **DWL**: Waiting because of G4 (only the rest of time is displayed in seconds)
- **MOV**: Moving of any axis (interpolator started)
- **POS**: Waiting in a position for signal
- **STOP**: Stop feeding state
- **0,0010**: Increment size 1
- **0,0100**: Increment size 10
- **0,1000**: Increment size 100
- **1,0000**: Increment size 1000



Attention! Each machine is different. Ask the machine tool builder about the marks of the scales and their exact values.

- **inch/mm**: Depends on the selection: mm or inch
- **F0%**: The override button is turned to 0%
- **F=0**: The programmed feed is 0

6 Sixth status field states

- **PLC**: Execution of PLC function is on
- **Dry Run**: Dry run

7 Seventh status field states

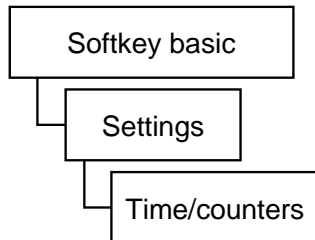
- *****: Scaling
- **@**: Rotation active
- **!**: The common zero shift is not zero
- **#**: Mirroring

8 General NC states

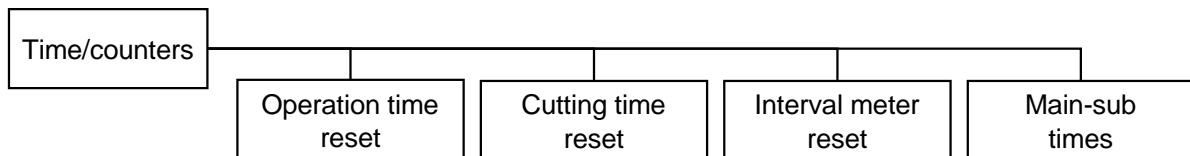
- **NC**: No **NC ready** signal
- **KLAV**: No reference point on any axis
- **EMG**: Emergency state
- **REF**: No reference point on any axis
- **TEST**: Test mode
- **MLOCK**: Machine lock
- **FLOCK**: Function lock
- **PLC**: PLC is not running

5.9. Part and Time Counter

This displays the different time and part counters. Apart from the operating time, the values are free to overwrite.



Time / counters	
Timers:	
Power on time:	2 : 00 : 51 : 03 . 385
Operation time:	0 : 02 : 29 : 36 . 834
Cutting time:	13 : 11 : 03 : 10 . 830
Interval meter:	0 : 00 : 12 : 00 . 000
Part time:	0 : 00 : 00 : 06 . 656
Counters: (M99)	
Parts count:	4
Parts require:	3
Parts total:	99
Parts today:	0



Power on time: Unmodifiable clock that begins when the machine is initially delivered.

Operation time: The time that the machine spends in automatic start state. It can be reset to zero by pushing the **Operation time reset** button in the **SOFTKEY** line.

Cutting time: The time that the machine is in feed movement and not in rapid traverse. It can be reset to zero by pushing the **Cutting time reset** button in the **SOFTKEY** line.

Interval meter: A timer for general use. It can be reset to zero by pushing the **Interval meter reset** button in the **SOFTKEY** line.

Part time: The part time counter measures the time spent in the automatic start state. This will be reset to zero automatically by starting a new part program.

Operation of the part counter

When starting machining of a series of parts, the **PARTS COUNT** counter must be set to 0. Set the **PARTS REQUIRE** value to the number of parts that will be produced. The **PARTS COUNT** counter is increased by one if the code M02 or M30 is read. The value of parameter N2305 **Part Count M** should be 0 for this counting mode.

If other M codes need to be used, (e.g. M99), set the above mentioned parameter value to the appropriate M code number, in this case to 99. If the number of machined parts reaches the number of parts to be produced, i.e. **PARTS COUNT** = **PARTS REQUIRED**, then the CNC will send a message to the operator that the parts count has reached the required number.





Further operation details are determined by the machine tool builder, see the supplementary manuals.

6. Program

6.1. Single block

SK > Program > Single block

With this function you can give an optional F (feed), S (spindle), T (tool), G (G-function), M (M-function) command or even execute a whole block containing one or more commands.

1. Press any of the buttons  (JOG),  (incremental JOG),  or hand wheel on the machine operation panel and the control unit will change the operation mode. The Single block window will be displayed on the left lower corner of the screen.
2. If you cannot find the window, you have to choose **Win-s** on the sidebar and push the **G0 X** button and the single block window will pop up every time.
3. In order to activate the window, click on to the top tab of the window and once the header is darkened, then the window is activated.
4. Type in the M3 S1000 X100 command line in any of the rows. You can even overwrite any of the previous command lines.
5. Push **ENTER** on the screen keypad and **SBLOCK** will appear on the status screen.
6. Press the **Cycle Start** button  and the block will be executed. The main spindle will start to rotate with 1000 RPM, then the slide will position to X=100 mm.

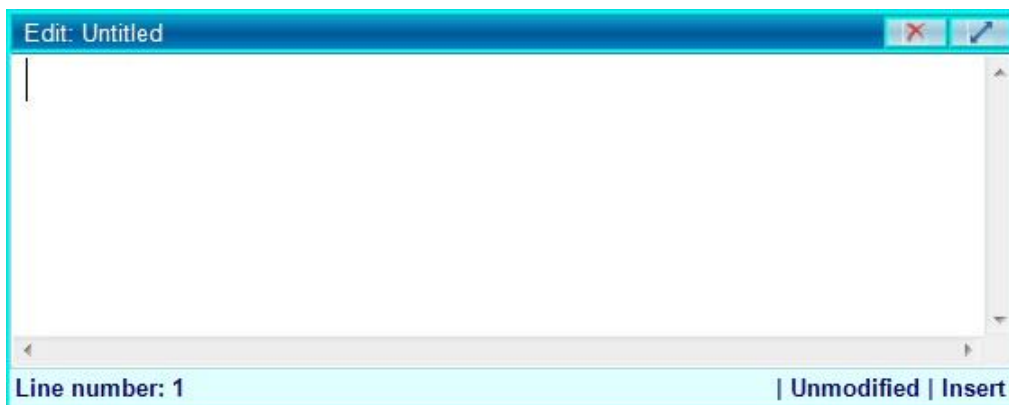
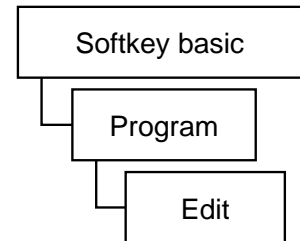


6.2. CNC Program

Creating and editing a CNC program can be done while the machine is still working. The program can be edited while another program is in operation. You can choose between writing a new program or editing a preexisting program.

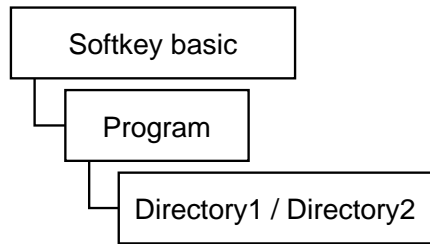
6.2.1. Creating a new CNC program

1. To create a new CNC program, select the **Edit** mode. A new window will pop up and **Edit: Untitled** will be on the header.
2. Turn on the keyboard tab on the right or the left side of the screen.
3. You can edit the program either by the virtual keypad or by the external keyboard connected to the USB port.
4. After the editing is finished, close the window (click on the red X on the header) and a message will show up, asking the following: “The text in the file has changed. Do you want to save the changes?”
5. Answer “Yes” then you can name the file and directory where you want to save the program, which can be done by pushing the **SAVE** button.
6. You can also save the program by pushing the **FILE** button on the **Softkey** line and then you can push the **Save** button to save the file immediately. If you select the **Save As** button, you can give a new name, or select an old file name which is already in the directory, which will then be overwritten with the new content.



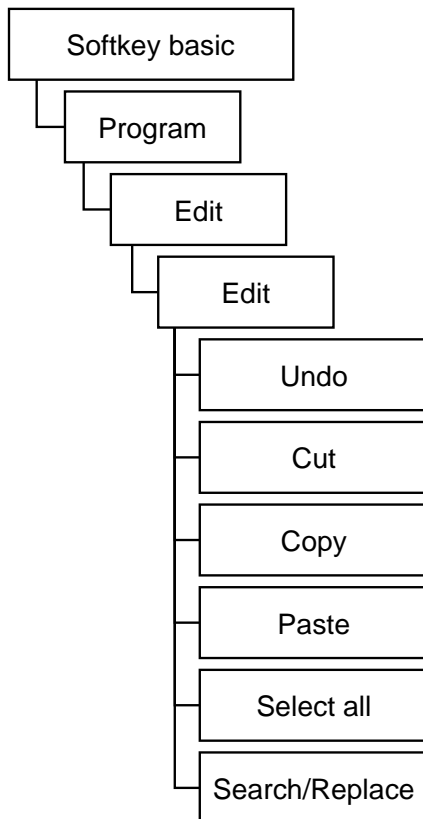
6.2.2. Editing an existing CNC program

1. To edit a program already existing, select the file first, which is possible by using one of the following command chains: **SK > Program > Directory1** or **SK > Program > Directory2**. This will bring up the directory window.
2. Search the part program that you wish to edit. You will be able to open only programs with .prg or .txt extensions.
3. Double-click on the desired program file and it will open up and you can start editing it.
4. After you are finished editing, click on the **File** button on the **Softkey** line. You can overwrite the original file by clicking on **Save** or you can also create a new file name by using **Save as**.
5. You can also save the file by clicking on the red X on the header of the window. The following message will show up in a window: "The text in the file has changed. Do you want to save the changes?" Push YES.



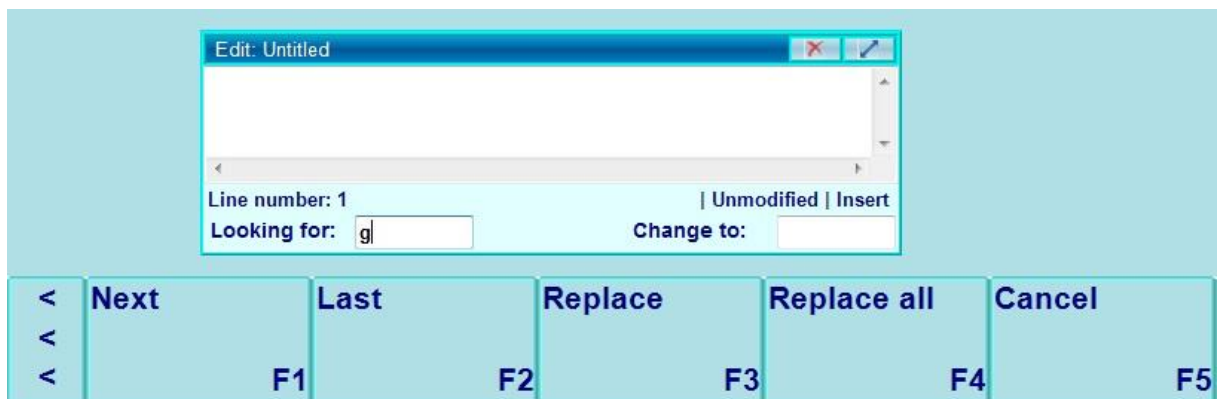
6. You can now give the file a new name and determine the directory where you want to save the program. Once completed, push the **SAVE** button to finalize.

6.2.3. Helping Functions by Editing



- **Undo:** This will undo the last operation.
- **Cut:** Select a part of or the whole text by highlighting it. Press the **Cut** button. The part you selected will disappear. The content you cut will be placed in a temporary storage and it will be copied from there to any place that you select when you press **Paste**.
- **Copy:** Select a part of or the whole text by highlighting it. Press **Copy** button. The text selected will be put in a temporary store without removing it from the main text. The content you copied will be placed in a temporary storage and it will be copied from there to any place that you select when you press **Paste**.
- **Paste:** The content of the temporary storage that is copied to the place selected by the cursor when you press **Paste**.
- **Select all:** The whole content of the editing window will be selected.
- **Search/Replace:** Two text fields will appear down in the editing window: **Looking for** and **Change to**. The content of **SOFTKEY** will

also be changed. Only the **Cancel** button will be active.



Use "**find**" field to locate any phrase that you are looking for within the program. The window will jump to the first result and select the line with the phrase. In case of several results, you can navigate among them by using the **Last** and **Next** buttons. You can replace any text that you put in the "**find**" field by using the **Replace** button once you've filled out the "**Change to**" text field.

6.2.4. Program Execution in Automatic operation mode

After you have created and edited a part program, you can select it to run in automatic mode.

1. To select the program to run, select one from either of the following directories: **SK > Program > Directory 1** or **Directory 2**
2. Move the cursor to the program file you wish to run and press **Run**.
3. Select **AUTO** mode in the machine



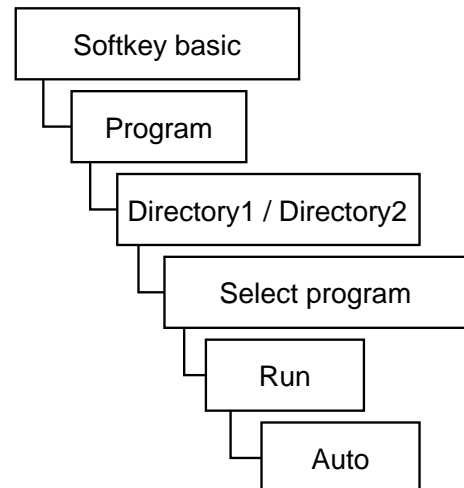
operation panel.

4. If there is a multi-channel machine, select the channel that will run the program and it will be loaded into the automatic execution storage field.



5. Push the **Cycle** start button and the program execution will start.

Note: If the machine operation panel of the screen is not activated, you can either change the operation mode on the panel of the machine tool or you have to activate the screen's panel.

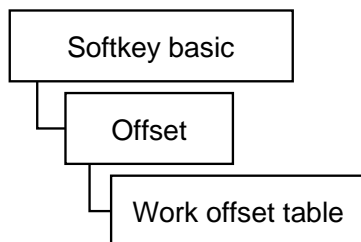


7. Offsets

7.1. 0-points (Work shift)

7.2. Work Offset

At the NCT201 control, you can use several different zero points. If the dimensions given in a drawing of a work piece cannot be programmed directly related to a basic point, the shifting of the program coordinates may help. You can save several zero points during the running of a part program and you can switch from one to another. You have the possibility to save all the zero points at the same time by assigning a "common" zero point. You can also rotate zero points.



Store the dimensions of the zero points in the following command chain:

SK > Offset > Work offset table

Work offset table							
[mm]	X offset	Y offset	Z offset	G17 r...	G18 r...	G19 r...	Comment
Common	0.000	0.000	0.000	0.000	0.000	0.000	
G52	0.000	0.000	0.000	---	---	---	---
G92	0.000	0.000	0.000	---	---	---	---
G54	0.000	0.000	0.000	0.000	0.000	0.000	
G55	0.000	0.000	0.000	0.000	0.000	0.000	
G56	0.000	0.000	0.000	0.000	0.000	0.000	
G57	0.000	0.000	0.000	0.000	0.000	0.000	
G58	0.000	0.000	0.000	0.000	0.000	0.000	
G59	0.000	0.000	0.000	0.000	0.000	0.000	
G54.1 P01	0.000	0.000	0.000	0.000	0.000	0.000	
G54.1 P02	0.000	0.000	0.000	0.000	0.000	0.000	
G54.1 P03	0.000	0.000	0.000	0.000	0.000	0.000	

Generally you can save six work shifts: **G54 G55 G56 G57 G58 G59**

The user can define an additional 99 work offsets:

G54.1 P01, G54.1 P02, G54.1 P03, G54.1 P04 ... G54.1 P99

G53 is the machine coordinate system that cannot make offset operations.

G54 is the default coordinate system of the machine that is selected after **RESET**.

7.2.1. Work Offset Measure

Work offset (work shift) measure is only possible in the following modes:

JOG, INCR, HNDL

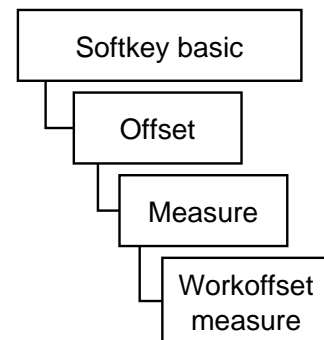
1. SK > Offset > Measure > Work offset measure

Machine position:		Measuring position:	
X +	0.000mm	X +	0.000mm
Y +	0.000mm	Y +	0.000mm
Z +	0.000mm	Z +	0.000mm

Work offs No:	Tool offs No:	Ref Axis:
G54	0	Z

N0 Q=0	D +	Geometry	Wear
		0.000mm	0.000mm

X +	Y +	Z +
0.000mm	0.000mm	0.000mm



2. Select the work offset (work shift) number. The coordinate system of the part program 0-point will be selected.

Double-click the darker field under: Work offs No:

Scroll down to the desired work offset number (G54-G59)

(the selected coordinate system will not be activated this time, it will be used only for the work offset measuring)

Work offs No: 1

G54

N0 Q=0 D +

3. Select the reference axis that will be used for the work shift measuring.



4) Zero-point metering on axes rectangular to the tool (usually X and Y):

- Along axis X (along the axes rectangular to the tool) take up a position by manual movement, which is well-defined for you. You can do this for example by touching the workpiece, for which you can use a raporter (gauge block), or a simple sheet of paper. For this you have to mill a clean surface! For starting the main spindle and setting its revolution number per minute, please use the Machine panel or the unique block window!

You can touch an existing basis surface, too, for example, one of the surfaces of the vice.

- After touching the (prepared) basis surface, please write to the “Metering position” the position in which the machine is, compared to the selected zero-point. If you would like to ensure that the current position of the machine is the X-direction origo of the requested zero-point, you have to write 0 into field X of the metering position. During the metering please take into account the tool diameter, as well! If you used a raporter or paper sheet for touching the surface, please take into account its thickness, too!

Offsets - Workoffset measure

Machine position:		Measuring position:	
X +	0.000mm	X	2.5mm
Y +	0.000mm	Y +	0.000mm
Z +	0.000mm	Z +	0.000mm

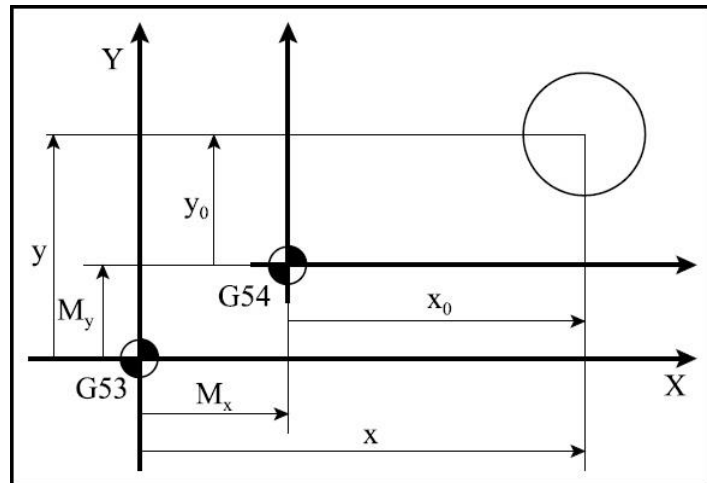
Work offs No: Tool offs No: 2 Ref Axis: Z

G55		Geometry		Wear	
N2 Q=0	D +	6.000mm	+		0.000mm
X +		0.000mm			
Y +		0.000mm			
Z +		0.000mm			
	L +	0.000mm	+		0.000mm

Then the control unit will automatically calculate the machine position of the zero-point from the machine position and the given metering position of the zero-point, based on the below equations:

$$M_x = x - x_0$$

$$M_y = y - y_0$$



where:

- M_x : the zero-point offset in direction X, the X-direction machine position of the origo of the zero-point
- M_y : the zero-point offset in direction Y, the Y-direction machine position of the origo of the zero-point
- x : the position of the tool measured in the machine coordinate system, in direction X
- y : the position of the tool measured in the machine coordinate system, in direction Y
- x_0 : the metering position entered for coordinate X, “The zero-point shall be situated in a way that the current position of the tool can be in the x_0 position of the zero-point.”
- y_0 : the metering position entered for coordinate Y, “The zero-point shall be situated in a way that the current position of the tool can be in the y_0 position of the zero-point.”

The modifications become recorded immediately also in the zero-point table.

You can see the machine position of the selected zero-point, thus the row of the zero-point table related to this zero-point. (Please see the frame in the below picture.)

Offsets - Workoffset measure			
Machine position:		Measuring position:	
X +	0.000mm	X +	0.000mm
Y +	0.000mm	Y +	0.000mm
Z +	0.000mm	Z +	0.000mm
Work offs No:		Tool offs No: 2	Ref Axis: Z
G55		Geometry	Wear
N2 Q=0	D +	6.000mm	+ 0.000mm
X +	0.000mm		
Y +	0.000mm		
Z +	0.000mm	L +	0.000mm + 0.000mm

- You can enter directly the machine position of the selected zero-point, if you have previously recorded it. (Then the control unit will automatically calculate the metering position from the machine position and the given machine position of the zero-point, which shows where the machine is situated interpreted from the selected zero-point).

Offsets - Workoffset measure

Machine position:		Measuring position:	
X +	0.000mm	X +	0.000mm
Y +	0.000mm	Y +	0.000mm
Z +	0.000mm	Z +	0.000mm

Work offs No: Tool offs No: **2** Ref Axis: **Z**

G55	Geometry		Wear	
N2 Q=0	D +	6.000mm	+	0.000mm
X		0.000mm		
Y +		0.000mm		
Z +		0.000mm		

The modifications entered here become recorded immediately in the zero-point table, as well.

5) Execute the steps described in point 3) also with axis Y (also with other axes rectangular to the other tools)!

6) Zero-point metering on axes falling into the direction of the tool (usually Z):

- If you know the tool length correction (as e.g. on an external measuring machine the metering of tools has already been performed, and the corrections have become recorded into the correction table), then you can submit it for the control unit: For this you have to select the tool correction number of the tool in the machine, which indicates that row of the tool correction table from which the control unit will take out the correction values, in order to take them into account during zero-point metering. (Double-click on the area circled in red, after which a writable text field will appear.)

Offsets - Workoffset measure

Machine position:		Measuring position:	
X +	0.000mm	X +	0.000mm
Y +	0.000mm	Y +	0.000mm
Z +	0.000mm	Z +	0.000mm

Work offs No: Tool offs No: **0** Ref Axis: **Z**

G55	Geometry		Wear	
N2 Q=0	D +	6.000mm	+	0.000mm
X +		0.000mm		
Y +		0.000mm		
Z +		0.000mm		

- If for the zero-point metering you would not like to use a tool, only the plain of the (empty) main spindle, you have to write a tool correction number with 0. This sets the length- and diameter geometry to zero. The yellow background number of the text field indicates that if

you forgot to rewrite the zero and still there is a tool of some size in the main spindle, it will lead to an erroneous zero-point metering, or it means that you entered such a high correction number that is bigger than the dimension of the tool correction table fixed on the parameter.

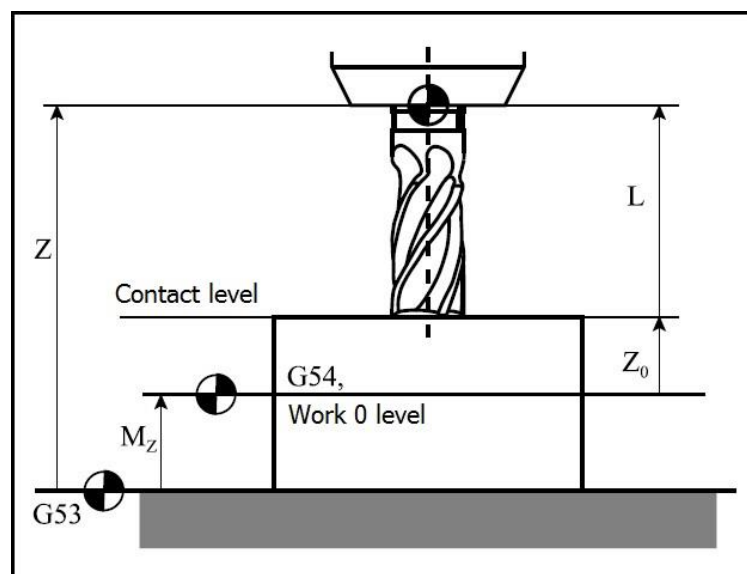
- Along axis Z (along the axis falling into the direction of the tool) take up a position by manual movement which is well-defined for you. You can do this for example by touching the table or the workpiece, for which you can use a raporter (gauge block) or a simple sheet of paper. For touching the workpiece you shall use a clean and milled surface! For starting the main spindle and setting its revolution number per minute, please use the Machine panel or the unique block window!

- After touching the (prepared) basis surface, please write to the “Metering position” the position in which the machine is, compared to the selected zero-point. If you would like to ensure that the current position of the machine is the Z-direction origo of the requested zero-point, you have to write 0 into field Z of the metering position. If you used a raporter or paper sheet for touching the surface, please take into account its thickness, too!

Offsets - Workoffset measure			
Machine position:		Measuring position:	
X +	0.000mm	X +	0.000mm
Y +	0.000mm	Y +	0.000mm
Z +	0.000mm	Z	2.5mm
Work offs No:		Tool offs No: 2	Ref Axis: Z
G55		Geometry	Wear
N2 Q=0	D +	6.000mm	+ 0.000mm
X +	0.000mm		
Y +	0.000mm		
Z +	0.000mm	L +	0.000mm +

Then the control unit will automatically calculate the machine position of the zero-point from the machine position and the given metering position of the zero-point, based on the below equations:

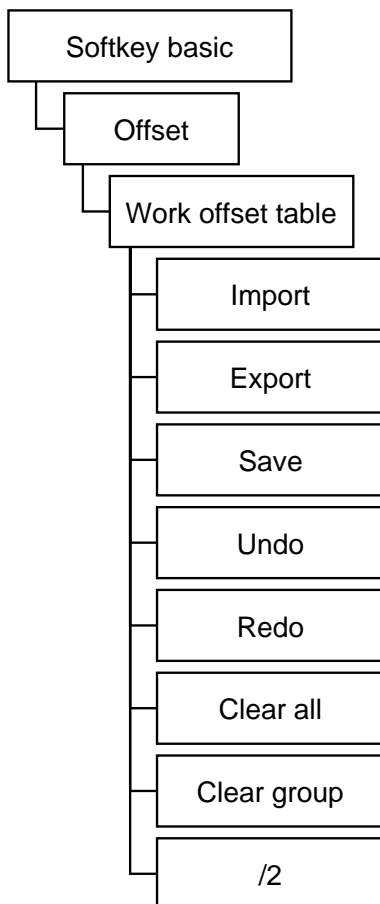
$$M_z = z - L - z_0$$



where:

- M_z : the zero-point offset in direction Z, the Z-direction machine position of the origo of the zero-point
 - z : the position of the tool measured in the machine coordinate system (without any length correction), in direction Z
 - L : The length correction of the tool (based on the submitted correction number.)
 - y_0 : the metering position entered for coordinate Z, “The zero-point shall be situated in a way that the current position of the tool can be in the z_0 position of the zero-point.”
- The modifications will be immediately registered in the work offset table.

7.2.2. Editing the Work Offset Table



Call the Work offset table through the following command chain:

SK > Offset > Work offset table.

To edit the table, touch the cell that you wish to overwrite and using an external keyboard or a keypad, press **ENTER**.

You can find the following editing functions in the **SOFTKEY** line:

Clear all: You can quickly cancel all values by using Clear all, which will cause all of the offset values to change to 0.

Clear group: You can quickly cancel all values of a selected work offset by using Clear group, which will cause all of the offset values to change to 0.

Save: You can save all the values in the offset table. The control unit will not be able to store more than one work offset table.

Export: If you wish to create several different work offset tables, you can export them by pressing the **Export** button. A directory will show up and you can type the name of the table and the place where you wish to save it.

Import: You can import a work offset table which was already saved in a different directory.


Undo: You can undo any previously made modifications. A window will pop up with all of the recent modifications and you simply need to select the one you wish to undo by using **Up** and **Down** buttons in the **SOFTKEY** line. After the selection is highlighted, press **Undo** button and an alarm window will pop up, asking you the question: “Do you really want to restore the selected workoffset/tooloffset values?” Press **Yes**.

Redo: You can cancel a previous undo operation by pressing **Redo** in **SOFTKEY** line. A window will pop up with the previous undo values. Select the modification you wish to redo by using the **Up** and **Down** buttons. After the selection is highlighted, press the **Redo** button and an alarm window will pop up, asking you

the question: "Do you really want to restore the selected workoffset/tooloffset values?" Press **Yes**.

/2: By this, for example, we can position a zero-point into the middle of an island, simply by touching the two sides of the island:

- 1) Touch one of the sides of the island!
- 2) Let's set the position of the given axis to 0 in the point of touch!
- 3) Touch the other side of the island!
- 4) Push the '/2' button in the second touch point. By this the zero-point will be set between the two touching points, in a way that the tool will be situated in the zero-point symmetrically, in the middle of the island.

***  **Attention!** Any operations made on work offset tables will immediately overwrite the offset register. The running program will take this into account only if you restart the program from the very beginning. You can also interrupt the program execution by hitting the **RESET** button. In this case, the coordinate system will be overwritten, but still the modified value won't be considered until the program will recall the modified coordinate system. If you make a careless overwriting of the coordinate system, it won't create an immediate problem during the program execution. The impact will only show up the next time that the program runs, so be very careful when you overwrite the work shift values.

7.2.3. Selection of work zero point in a program

See NCT 201 program description for more details.

7.3. Tool Offsets

The tool offset values are used for the tool length and tool nose radius compensation in the CNC program. The tool offsets are collected in the offset tables. Each time a tool offset is called by the program, the data of the selected offset will be used in the program. The programmed path of the tool will be modified by the offset values.

Beyond the correction, it is possible to keep register of the wear of the tool, too, in a separate compartment.

In case of a milling machine, the tool correction and wear are tool characteristics belonging together, and we can refer to them jointly, by the correction number.

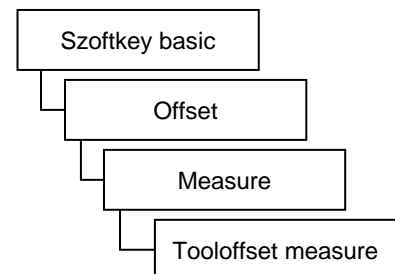
7.3.1. Tool offset measuring (inside the milling machine)

The tool offset can be measured in the following operation modes: **JOG, INCR, HNDL**

1 SK > Offset > Measure > Tooloffset measure

2. Select the work offset (work shift) number. The coordinate system of the 0 point part program will be selected.

Double-click the darker field under **Work offs No.** Scroll down to the desired work offset number (G54-G59). The selected coordinate system will not be activated this time, it will only be used for the tool offset measuring.



Machine position:		Measuring position:	
X -	6323.8529 mm	X -	6756.8389 mm
Y -	9453.5077 mm	Y -	9678.3214 mm
Z -	955.3714 mm	Z -	1221.1071 mm

Work offs No:	Tool offs No:	Ref Axis:
G54	0	Z

	Geometry	Wear
G54	+ 0.0000 mm	+ 0.0000 mm
G55		
G56		
G57		
G58	L + 0.0000 mm	+ 0.0000 mm
G59		
G54.1 P01		
G54.1 P02		
G54.1 P03		
G54.1 P04		
G54.1 P05		

3. Select the tool offset number that will be used during the work offset measuring then double click on the field right to the Tool offset No and type in the offset number and press **ENTER**.

Offsets - Tooloffset measure

Machine position:		Measuring position:	
X +	0.000 _{mm}	X +	0.000 _{mm}
Y +	0.000 _{mm}	Y +	0.000 _{mm}
Z +	0.000 _{mm}	Z +	0.000 _{mm}
Work offs No:	Tool offs No: 1	Ref Axis:	Z
G54	Geometry	Wear	
N0 Q=0	D +	0.000 _{mm}	+ 0.000 _{mm}
X +	0.000 _{mm}		
Y +	0.000 _{mm}		
Z +	0.000 _{mm}	L +	0.000 _{mm} + 0.000 _{mm}

If you select an invalid offset number (which is not in the offset table), the field background will be yellow. The control will use the stored offset values of the tool from the selected offset number. The 4 values are: **length**, **tool tip radius**, **wear offset** and **tool radius location** (this is an imaginary tool tip locator)

4. Select the reference axis for the tool offset measure.

122 1.107 1 mm

Ref Axis: Z

Wear X

+ 0.000 Y

Z

5) Execute the following steps:

- ❖ Directly under the labels “Geometry” and “Wear”, you have to enter the nominal diameter and wear of the tool. (Double-click on the area circled in red, after which a writable text field will appear.)

Machine position:		Measuring position:	
X +	0.000mm	X +	0.000mm
Y +	0.000mm	Y +	0.000mm
Z +	0.000mm	Z +	0.000mm
Work offs No: G54		Tool offs No: 1	Ref Axis: Y
		Geometry	Wear
N1 Q=0	D +	6.000mm +	0.000mm
X +	0.000mm		
Y +	0.000mm	L +	0.000mm +
Z +	0.000mm		

- ❖ The determination of the length correction has to be carried out only on the axis parallel to the tool - on the selected reference axis.



Example: Determination of length correction on axis Z:

For determining the length correction you can select from two options:

- Direct determination of length correction: If you know the exact length of the tool (as, for example, the metering of the tool has already been performed on an external metering machine), you can enter it into the place corresponding to the selected reference axis, at the bottom of the window, after “L+”.

Machine position:		Measuring position:	
X +	0.000mm	X +	0.000mm
Y +	0.000mm	Y +	0.000mm
Z +	0.000mm	Z +	0.000mm
Work offs No: G54		Tool offs No: 1	Ref Axis: Y
		Geometry	Wear
N1 Q=0	D +	6.000mm +	0.000mm
X +	0.000mm		
Y +	0.000mm	L +	0.000mm +
Z +	0.000mm		

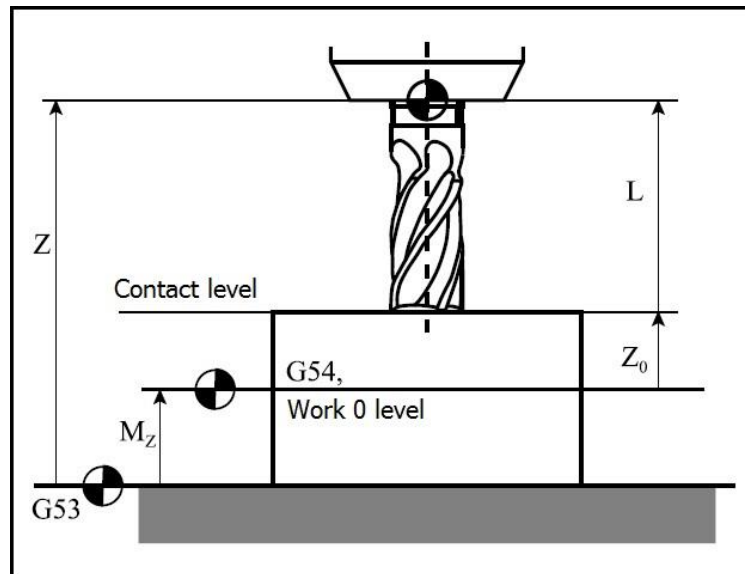
- Metering of the length correction: Along the selected reference axis take up a position by the tool which is well-defined for you.
Then you can mill a clean basis surface in the held workpiece, or you can touch an existing surface. For this you can use a raporter (gauge block), or a simple sheet of paper. For starting the main spindle and setting its revolution number per minute, please use the Machine panel or the unique block window!

After touching the (prepared) basis surface, please write to the “Metering position” the position in which the machine is, compared to the selected zero-point. If you would like to ensure that the current position of the tool is the origo of the requested zero-point, you have to write 0 into field Z of the metering position. If you used a raporter or paper sheet for touching the surface, please take into account its thickness, too!

Offsets - Tooloffset measure			
Machine position:		Measuring position:	
X +	0.000mm	X +	0.000mm
Y +	0.000mm	Y +	0.000mm
Z +	0.000mm	Z	2.5mm
Work offs No:		Tool offs No: 2	Ref Axis: Z
G55		Geometry	Wear
N2 Q=0	D +	6.000mm	+ 0.000mm
X +	0.000mm		
Y +	0.000mm		
Z +	0.000mm	L +	0.000mm + 0.000mm

Based on the machine position, the control unit will see where the tool is situated in the selected zero-point; by the entered “Metering position:” we can determine where it “should” be in the selected zero-point. Based on all these the control unit is going to calculate the elongation of the tool, according to the below equation:

$$L = Z - Z_0 - M_z$$



where:

- M_z : the zero-point offset in direction Z , the Z -direction machine position of the origo of the zero-point
- Z : the position of the tool reference point measured in the machine coordinate system (without any length correction), in direction Z
- L : length correction calculated by the control unit
- Z_0 : the metering position entered for coordinate Z , “The length correction of the tool shall be that big that the current position of the tool can be in the z_0 position of the zero-point.”

The modifications become recorded immediately also in the tool correction table.

7.3.2. Editing the Tool Offset Table

Call the work offset table through the following command chain:

SK > Offset > Tool offset table.

To edit the table, touch the cell that you wish to overwrite and press **ENTER** using an external keyboard or keypad.

[mm]	L geometry	L wear	D geometry	D wear	Q value	Comment
N1	0.000	0.000	0.000	0.000	0	No comment
N2	0.000	0.000	0.000	0.000	0	No comment
N3	0.000	0.000	0.000	0.000	0	No comment
N4	0.000	0.000	0.000	0.000	0	No comment
N5	0.000	0.000	0.000	0.000	0	No comment
N6	0.000	0.000	0.000	0.000	0	No comment
N7	0.000	0.000	0.000	0.000	0	No comment
N8	0.000	0.000	0.000	0.000	0	No comment
N9	0.000	0.000	0.000	0.000	0	No comment

You can find the following editing functions in the **SOFTKEY** line:

Clear all: You can quickly cancel all values by using Clear all, which will cause all of the offset values to change to 0.

Clear group: You can quickly cancel all values of a selected tool offset by using Clear group, which will cause all of the offset values to change to 0.

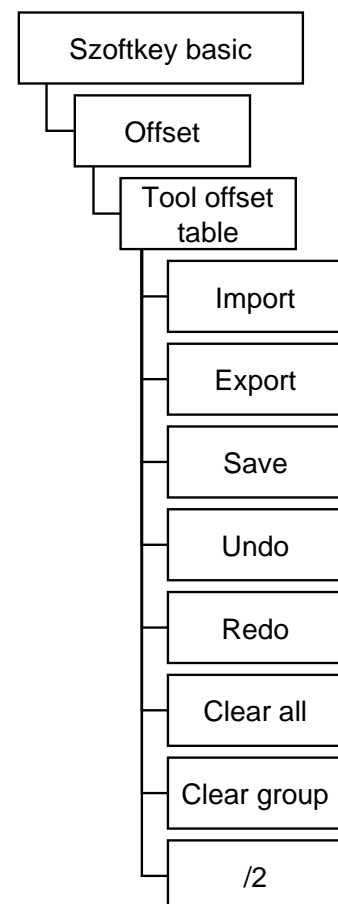
Save: You can save all the values in the offset table. The control unit will not be able to store more than one work offset table.

Export: If you wish to create several different tool offset tables, you can export them by pressing the **Export** button. A directory will show up and you can type the name of the table and the place where you wish to save it.

Import: You can import a tool offset table which was already saved in a different directory.

Undo: You can undo any previously made modifications. A window will pop up with all of the recent modifications and you simply need to select the one you wish to undo by using **Up** and **Down** buttons in the **SOFTKEY** line. After the selection is highlighted, press **Undo** button and an alarm window will pop up, asking you the question: "Do you really want to restore the selected workoffset/tooloffset values?" Press **Yes**.

Redo: You can cancel a previous undo operation by pressing **Redo** in SOFTKEY line. A window will pop up with the previous undo values. Select the modification you wish to redo by using the **Up** and **Down** buttons. After the selection is highlighted, press the **Redo** button and an alarm window will pop up, asking you the question: "Do you really want to restore the selected workoffset/tooloffset values?" Press **Yes**.





Attention! Any operations made on work offset tables will immediately overwrite the offset register. The running program will take this into account only if you restart the program from the very beginning. You can also interrupt the program execution by hitting the **RESET** button. In this case, the coordinate system will be overwritten, but still the modified value won't be considered until the program will recall the modified coordinate system. If you make a careless overwriting of the coordinate system, it won't create an immediate problem during the program execution. The impact will only show up the next time that the program runs, so be very careful when you overwrite the work shift values.



Note: The correction registers with the same number constitute the same group, although in the program we have to refer to the diameter (D or R) length corrections (L) separately after the ordinal number following D and H letters.

7.4. Orientation and Offset of the main spindle

With this function you can set the orientation angle of the spindle. This is the angle position to which the spindle will rotate upon the **M19** command. If it is a multi-spindle machine, the orientation angle can be adjusted for each of the spindles separately.

7.4.1. Setting the orientation position

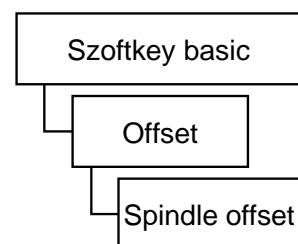
Spindle	M3-M4	Ref	Ch	M11-18	Angle	Impulse	Phas	Param
Spindle		<input type="checkbox"/>						

Calculation value: 0 [Imp.] Calculation + offset

Offset value: 0 [°] Set offset

Offset in angle ☒

1. **SK > Offset > Spindle offset.**
2. In the **Phase Shift Calculation** pop up window, select **Orienting position calculation** in the drop down menu next to the Type line.
3. Double click in the first cell of the table in the Spindle line and a drop down menu will appear.
4. Select the spindle orientation angle you wish to adjust.
5. Next to the Calculation value line, you will find the current position of the spindle expressed in impulses. You can choose between the following two options:



- You can give the absolute angle position of the spindle if you know the null-angle position (locate this in the manual or contact the machine tool builder). Input the absolute angle position next to the **Offset value**, then check the small box next to the **Offset in angle** line and press **Set offset**. The spindle will orient to the angle set when you use the **M19** command.
- You can rotate the spindle in a position you know (i.e. aligning the flat surface of a tool parallel to an axis) as a reference point. To set a certain angle, turn the tool in the angle that you wish. Input the angle of the rotation next to **Offset value**, then check the small box next to the **Offset in angle** line and press **Set offset**. The spindle will orient to the angle set when you use the **M19** command.



Suggestion: You can give the offset in increments as well if you know the resolution of the encoder of the spindle.

7.4.2. Driving Spindles in Shifted Phase Mode (part transfer between spindles)

This function makes it possible to rotate two spindles of a multi-spindle machine in different phases with shifted angles, but with equal revolutions. It can be useful in a sub-spindle machine, where you wish to transfer a part from one spindle to another. In this case, it is not enough to have just the same revolutions for a secure transfer. The opposite side spindle can have a fixture which can clamp the part only in a certain phase.

7.4.2.1. Calculation of Phase Offset

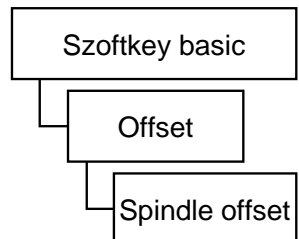
	Spindle	M3-M4	Ref	Ch	M11-18	Angle	Impulse	Phas	Param
Master spin...			<input type="checkbox"/>						
Slave spindle			<input type="checkbox"/>						

Calculation value: 0 [Imp.] Calculation + offset

Offset value: 0 [°] Set offset

Offset in angle ☒

1. **SK > Offset > Spindle offset.**
2. In the pop up window (pictured above) select **Phase shift calculation** in the drop down menu next to the **Type** line.
3. Double click in the first cell of the table (in the **Master spindle** line and the **spindle** column) and a drop down menu will appear.
4. Select the spindle orientation angle you wish to adjust.
5. You can select which spindle shall be the master spindle and the slave spindle. The two spindles will rotate in a set shifted phase.
6. If you select a slave spindle, you can find the momentary spindle position. To calculate the phase offset, you can choose between two solutions:
 - You can give the phase offset of the slave spindle compared to the master spindle. To do this, Input the absolute angle position next to **Offset value**, check the window **Offset in angle**, then press **Set offset**. Using command M22, the slave spindle will take the same RPM as the master spindle. The two spindles will rotate with the given offset phase. If you use the M3 or M4 commands for the slave spindle, it will have no effect. It can be stopped only by the M5 command, which will also finish the synchronization of the spindles.
 - Rotate the spindles in the desired positions and then you can give an angle value of your choice. If you input 0 degrees, the spindles will turn together in the position you just set manually. Input the angle value to the **Offset value**, then check the **Offset in angle** window and press **Calculate + offset**. The control will be able to calculate the phase shift automatically from the angle value. Using the M22 command, the slave spindle will take the same RPM as the



master spindle. The two spindles will rotate with the given offset angle. If you use the M3 or M4 commands for the slave spindle, it will have no effect. It can be stopped only by the M5 command, which will also finish the synchronization of the spindles.



Suggestion: You can give the offset in increments as well if you know the resolution of the encoder of the spindle.

8. Tool management

Definitions: For the storage of tools we use a *tool storage* (e.g. chain/drum/matrix storage). In case of several storages we call the separate storages as *magazines*. Within a magazine there are *pockets* - tool pockets situated. The tools can be placed into the pockets.

The NC 201 control unit provides the below tool management possibilities:

- The situation of tools in the store can be different (random storage) or fix (the tool always gets back to the same pocket).
- From the aspect of calls from a part program, the tool register can be place-coded or tool-coded. In the latter case the tool code is the so called "T code".
- The control unit is able to manage the so called *oversized* tools occupying more pockets. All this in an asymmetric way, with an accuracy of half pocket. The direction-dependent half-pocket register has space-saving reasons: If a tool hangs over to an extent of a half-pocket to the next pocket, no tool can be placed into it; however, e.g. two tools which are oversized to an extent of half pockets each can be placed next to each other by leaving out a pocket.
- The control unit is able to measure and keep records of the tool life of tools. In case of a tool-coded tool register, the interchangeable tools shall be organized into *groups*. In such case we refer to the group and all tools by the same T code.

8.1. Adding a new tool to the store

It is possible to add tools only in the editing mode.



Note: The exact procedure of adding a tool is determined by the machine constructor, the tool addition presented here is only an example, as it is done typically in this way; however, your machine can be different from this. Please contact the machine constructor for more details about the procedure of adding a tool.

First you have to place the tool into the machine! The placement of the tool can be done directly into the store, or through a spindle- or standby storage. The placement of the tool through a spindle may take place e.g. in the following way:

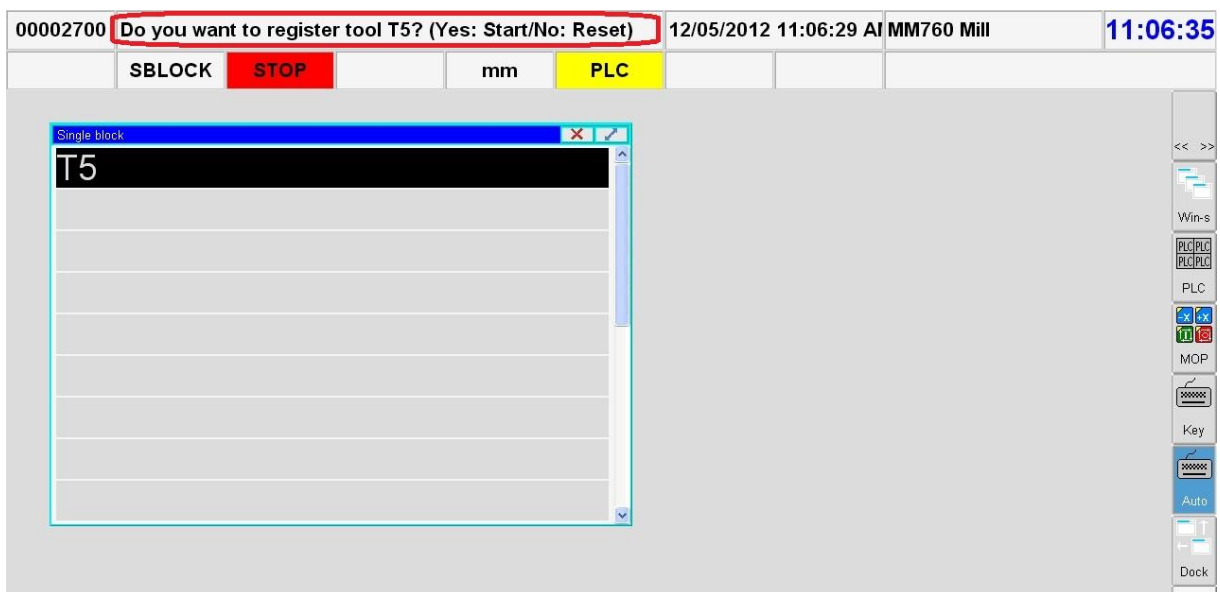
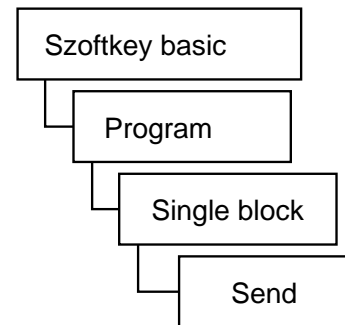


Example: Push the “T reg” (T registration) button on the machine control panel! The button will stuck in (its background starts to light).

In the base state of the SOFTKEY select the menu option “Program/Unique block”!

If you would like to define a tool which will not be a member of any existing tool groups, you have to write a not-yet-occupied T code, too in to the unique block window. (e.g. T1) If you would not like to extend the existing tool group with a new tool, you have to enter the T code of the group to be extended. Execute the unique block! (Push the “Send” button on the Softkey row, then the START button on the machine control panel.)

The control unit will get into the Stop state and in a message it will ask whether or not to register the T tool?



Push the START button!

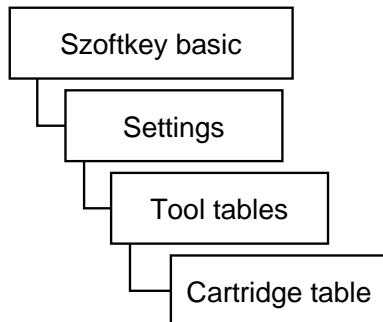
In case of a tool addition through the main spindle, if there is another tool in the main spindle, the control unit will first place it into the store, then ask for the placement of the tool to be registered into the spindle.



According to the design of the machine, place the tool into the main spindle then push again the START button.

Then the control unit is going to warn in a message not to forget to enter data belonging to the tool: the correction values, technical parameters, the shape number, data related to tool life and other data.

00005100	Fill in columns tool info and figure No.	12/05/2012 11:09:41 AM	MM760 Mill	11:10:38
	SBLOCK	STOP	mm	PLC



For this select the menu option “Settings/Tool tables/Tool place table” in the base state of the SOFTKEY.

In the opening “Tool storage table” we can see that the placed-in tool got into a special one-pocket magazine, the main spindle. Double-touch on the T code in the “data number” column of the pocket storing the tool.

00005100	Fill in columns tool info and figure No.	12/05/2012 11:09:41 AM	MM760 Mill	11:14:30
	SBLOCK	STOP	mm	PLC

Row	Cartridge	Pot	ToolTable Row	Tool Fi...
1	1	1		
2	1	2	T10 [4]	0
3	1	3	T2 [2]	0
4	1	4	T1 [1]	0
5	1	5		
6	1	6		
7	1	7		
8	1	8		
9	1	9		
10	1	10		
11	1	11		
12	1	12	T3 [3]	0
13	1	13		
14	1	14		
15	1	15		
16	1	16		
17	S Spindle [10]		T5 [5]	0

Upon this the “Tool edition” window will appear.

First select the “Edit” button located in the middle of the appearing window.

The screenshot shows a window titled "SingleToolEdit Channel1". It contains several sections of input fields:

- Tool Row Id.:** Cartridge: 10, Pot: 1
- ToolType Num.:** 5
- NickName:** (empty field)
- Tool info:** UEFI, **Edit** (circled in red)
- Tool Fig. No.:** 0 [0;0] (dropdown menu)
- Left dir.:** 0, **Right dir.:** 0
- Life Status:** Disabled (dropdown menu)
- Life Cnt:** 0
- Life Max:** 0
- Life Notice:** 0
- H index:** 0
- Length geom. (mm):** 0
- Length wear (mm):** 0
- D index:** 0
- D geom. (mm):** 0
- D wear (mm):** 0
- Q value:** 0
- Spindle:** 0
- Feed:** 0

At the bottom right, there are two buttons: a green checkmark and a red X.

Upon this the following window will appear:

The screenshot shows a window titled "Tool information edit". It contains several sections of radio button options:

- Data is:** ☐ [I] Invalid, ☒ [V] Valid
- Life count:** ☒ [C] Count, ☐ [T] Time
- Tool size:** ☒ [N] Normal, ☐ [L] Large
- Data edit:** ☒ [E] Enabled, ☐ [D] Disabled
- Tool is:** ☒ [U] Unsearch, ☐ [S] Search

At the bottom, there are two buttons: a green checkmark and a red X.

The following shall be set here:

- Data series: We use the “Invalid” function for declaring a tool as unused without taking it out from the machine.
- Tool life calculation: If we would like to use the tool life management, we have to select whether we would like to measure the “occasion” of redemptions of the tool or the “time” spent with tool cutting.

- Tool size: If the tool fits into a place of a pocket, let's select the "normal" option. If it overhangs by a half-pocket to any direction, we have to select the "oversized" option. (The exact setting of elongation is described in the chapter "Oversized tool".)
- Data series can be edited: By this we can set whether the data belonging to the tool can be edited or not. If a tool is set as uneditable, e.g. the PLC cannot rewrite the data series of the tool neither.
- NC can look for the following: If we would like to ensure that this tool can be called in the NC program as a change tool, during the change of an interchangeable tool (as e.g. the tool life of the other has expired), we have to select the option "Yes".

After this we have to click on the green tick at the bottom of the window and return to the previous window.

If you have already metered the tool from outside, on the right side of the window (e.g. in case of a milling machine) you can fill in the L geometry (Length geometry), L wear, D geometry (Diameter geometry), D wear and other values.

On the right side, too, at the bottom of the window you can enter the technological values (Main spindle revolution number per minute, Feed) belonging to the tool. These can be called from an NC program.

We can set the tool life management in the left bottom corner of the window. For the time being you have to select the "No" option in the scroll-down menu "Tool life status".

Life Status	Disabled
Life Cnt	Disabled
Life Max	Not used
Life Notice	Remains
	Expired
	Breakage
	<input type="checkbox"/>

For details please see the chapter “Tool life management”.

Set the shape number of the tool if it is oversized.

For details please see the chapter “Oversized tool”.

Finally push again the START button, upon which the message “Fill in the tool info and shape number window!” will disappear.

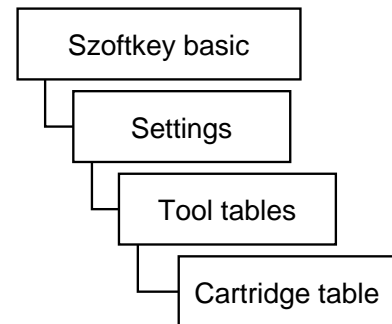
Switch off the “T reg” button on the machine control panel!

8.2. Tool life management:

In the base state of the SOFTKEY select the menu option “Settings/Tool tables/Tool place table”. In the opening “Tool place table” you have to select a tool the tool life of which you would like to manage; then double-touch on the T code of the tool located in the data number column.

Cartridge management table				
Row	Cartridge	Pot	ToolTable Row	Tool Fi...
1	1	1		
2	1	2	T10 [4]	0
3	1	3	T2 [2]	0
4	1	4	T1 [1]	0
5	1	5		
6	1	6		
7	1	7		
8	1	8		
9	1	9		
10	1	10		
11	1	11		
12	1	12	T3 [3]	0
13	1	13		
14	1	14		
15	1	15		
16	1	16		
Row	Cartridge	ToolTable Row	Tool Fi...	
17	S Spindle [10]	T5 [5]	0	

Select the “Edit” button located in the middle of the appearing window.



SingleToolEdit Channel1

Tool Row Id.: Cartridge: 10 Pot: 1		H index <input type="text" value="0"/>
ToolType Num. <input type="text" value="5"/>	Length geom. (mm) <input type="text" value="0"/>	Length wear (mm) <input type="text" value="0"/>
NickName <input type="text"/>	D index <input type="text" value="0"/>	D geom. (mm) <input type="text" value="0"/>
Tool info <input type="button" value="UENCI"/> <input type="button" value="Edit"/>	D wear (mm) <input type="text" value="0"/>	Q value <input type="text" value="0"/>
Tool Fig. No. <input type="text" value="0 [0;0]"/>	Left dir. <input type="text" value="0"/>	Right dir. <input type="text" value="0"/>
Life Status <input type="text" value="Disabled"/>	Spindle <input type="text" value="0"/>	Feed <input type="text" value="0"/>
Life Cnt <input type="text" value="0"/>	<input type="button" value="✓"/> <input type="button" value="✗"/>	
Life Max <input type="text" value="0"/>		
Life Notice <input type="text" value="0"/>		

Upon this the following window will appear:

Tool information edit

Data is:	<input type="radio"/> [I] Invalid	<input checked="" type="radio"/> [V] Valid
Life count:	<input checked="" type="radio"/> [C] Count	<input type="radio"/> [T] Time
Tool size:	<input checked="" type="radio"/> [N] Normal	<input type="radio"/> [L] Large
Data edit:	<input checked="" type="radio"/> [E] Enabled	<input type="radio"/> [D] Disabled
Tool is:	<input checked="" type="radio"/> [U] Unsearch	<input type="radio"/> [S] Search
<input type="button" value="✓"/> <input type="button" value="✗"/>		

Select the appropriate option in the “Tool life counting” row:

- - For occasion: it measures the “occasions” of tool redemption.
- - For time: it measures the “time” of the tool spent by cutting.

After this we have to click on the green tick at the bottom of the window and return to the previous window, in the left bottom corner of which we can set the tool life management of the tool.

Life Status	Disabled ▼
Life Cnt	0
Life Max	0
Life Notice	0

- Tool life counter: This measures the time of the tool spent by cutting, or the number of its redemptions.
- Tool life: This is the length of tool life of the tool. If the value of the tool life counter reaches the tool life of the tool, the tool will be regarded as used, “Expired”.
- Warning tool life: in case the tool life counter reaches the value of the warning tool life, the control unit will indicate it in a message that the tool life of the given tool will soon expire.

The following states can be found in the scroll-down menu “Tool life status” with the below meanings:

Life Status	Disabled ▼
Life Cnt	0
Life Max	0
Life Notice	0

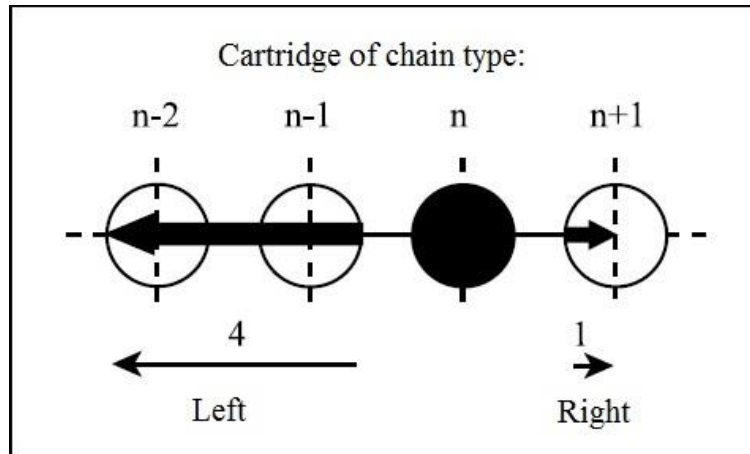
- No: We do not use the tool life management.
- Not used: The value of the tool life counter is 0, the tool has not been used yet.
- Expired: If the value of the tool life counter reaches the tool life of the tool, the tool will be regarded as used, “Expired”.
- Used: Then the tool life of the tool is not zero any more, but it has not expired either.
- Broken: This state can be set only by the machine operator, thus it can be seen that the tool cannot be used but it does not have to be taken out immediately from the storage.

The control unit will automatically change the states Not used, Used, Expired - by the increasing of occasions or the time spent by cutting - however, states Broken and No shall be set here by the operator of the machine.

After this we have to click on the green tick at the bottom of the window and the modifications will be saved.

8.3. Oversized tool

The control unit is able to manage the so called *oversized* tools occupying more pockets. All this in an asymmetric way, with an accuracy of half pocket. The oversize of tools, in case of a chain and drum store can be determined in one dimension (to the left or to the right);

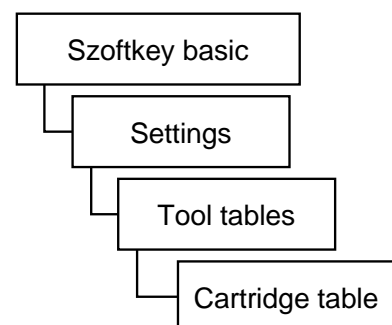


These tool characteristics (elongations to the left or to the right) are collected in a so called “tool shape table”. The below picture shows an example of the tool shape table of a chain storage machine:

Oversize Tool pattern table		
Fig. Num.	Left dir.	Right dir.
1	0	1
2	0	2
3	1	1
4	2	2
5	0	0
6	0	0
7	0	0
8	0	0
9	0	0
10	0	0
11	0	0
12	0	0
13	0	0
14	0	0
15	0	0
16	0	0
17	0	0
18	0	0
19	0	0
20	0	0

A so called “shape number” belongs to each tool. This shape number points to a row of the tool shape table, in which we can find the oversizes belonging to the given shape number. Before determining the oversizes of tools we have to select a shape number where we would like to save the oversized of the given tool. After this it is necessary to fill out the selected shape number columns in the tool shape table. Next, during the tool changes the control unit will manage the tool by taking into account the oversizes determined by the shape number.

In the base state of the SOFTKEY select the menu option “Settings/Tool tables/Tool place table”. In the opening “Tool place table” you have to select a tool you would like to set as oversized; then double-touch on the T code of the tool located in the data number column.



Cartridge management table				
Row	Cartridge	Pot	ToolTable Row	Tool Fi...
1	1	1		
2	1	2	T10 [4]	0
3	1	3	T2 [2]	0
4	1	4	T1 [1]	0
5	1	5		
6	1	6		
7	1	7		
8	1	8		
9	1	9		
10	1	10		
11	1	11		
12	1	12	T3 [3]	0
13	1	13		
14	1	14		
15	1	15		
16	1	16		
Row	Cartridge	ToolTable Row	Tool Fi...	
17	S Spindle [10]	T5 [5]	0	

Select the “Edit” button located in the middle of the appearing window.

SingleToolEdit Channel1	
Tool Row Id.: Cartridge: 10 Pot: 1	
ToolType Num. <input type="text" value="5"/> NickName <input type="text"/> Tool info <input type="button" value="UE/CI"/> <input type="button" value="Edit"/>	H index <input type="text" value="0"/> Length geom. (mm) <input type="text" value="0"/> Length wear (mm) <input type="text" value="0"/> D index <input type="text" value="0"/> D geom. (mm) <input type="text" value="0"/> D wear (mm) <input type="text" value="0"/> Q value <input type="text" value="0"/>
Tool Fig. No. <input type="text" value="0 [0;0]"/> <input type="button" value="v"/> Left dir. <input type="text" value="0"/> Right dir. <input type="text" value="0"/>	
Life Status <input type="text" value="Disabled"/> <input type="button" value="v"/> Life Cnt <input type="text" value="0"/> Life Max <input type="text" value="0"/> Life Notice <input type="text" value="0"/>	Spindle <input type="text" value="0"/> Feed <input type="text" value="0"/>
<div style="display: flex; justify-content: space-around;"> <input type="button" value="✓"/> <input type="button" value="✗"/> </div>	

Upon this the following window will appear:



Tool information edit

Data is: ☐ [I] Invalid ☒ [V] Valid

Life count: ☒ [C] Count ☐ [T] Time

Tool size: ☒ [N] Normal ☐ [L] Large

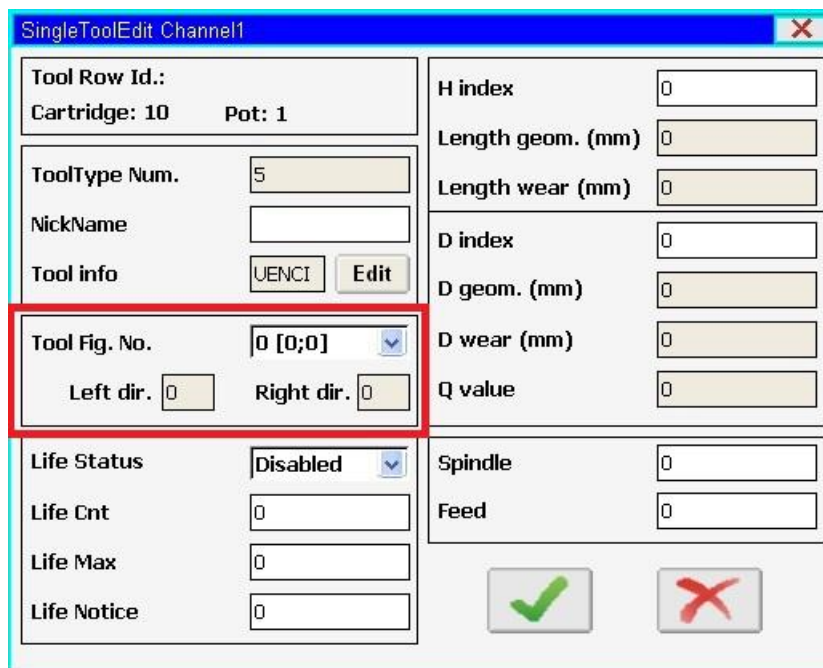
Data edit: ☒ [E] Enabled ☐ [D] Disabled

Tool is: ☒ [U] Unsearch ☐ [S] Search



 

In the “Tool size” row we have to select the “Oversized” option.

After this we have to click on the green tick at the bottom of the window and return to the previous window, on the left side of which we can set the oversize of the tool.





SingleToolEdit Channel1

Tool Row Id.: Cartridge: 10 Pot: 1		H index <input type="text" value="0"/>
ToolType Num. <input type="text" value="5"/>	NickName <input type="text"/>	Length geom. (mm) <input type="text" value="0"/>
Tool info <input type="text" value="UENCI"/> <input type="button" value="Edit"/>		Length wear (mm) <input type="text" value="0"/>
Tool Fig. No. <input type="text" value="0 [0;0]"/> <input type="button" value="v"/>		D index <input type="text" value="0"/>
Left dir. <input type="text" value="0"/> Right dir. <input type="text" value="0"/>		D geom. (mm) <input type="text" value="0"/>
		D wear (mm) <input type="text" value="0"/>
		Q value <input type="text" value="0"/>
Life Status <input type="text" value="Disabled"/> <input type="button" value="v"/>		Spindle <input type="text" value="0"/>
Life Cnt <input type="text" value="0"/>		Feed <input type="text" value="0"/>
Life Max <input type="text" value="0"/>		
Life Notice <input type="text" value="0"/>		
		 

In the “Shape number” field in the scroll-down menu you have to select a shape number, which contains the overhanging of the tool, or where you would like to save it.

SingleToolEdit Channel1

Tool Row Id.: Cartridge: 10 Pot: 1		H index 0
ToolType Num. 5		Length geom. (mm) 0
NickName		Length wear (mm) 0
Tool info UENCI Edit		D index 0
Tool Fig. No. 0 [0;0]		D geom. (mm) 0
Left dir. 0		D wear (mm) 0
Life Status		Q value 0
Life Cnt		Spindle 0
Life Max		Feed 0
Life Notice		 

0 [0;0]
 1 [0;1]
 2 [0;2]
 3 [1;1]
 4 [2;2]
 5 [0;0]
 6 [0;0]
 7 [0;0]
 8 [0;0]
 9 [0;0]
 10 [0;0]
 11 [0;0]
 12 [0;0]
 13 [0;0]
 14 [0;0]
 15 [0;0]
 16 [0;0]
 17 [0;0]
 18 [0;0]
 19 [0;0]
 20 [0;0]

Behind the tools we can see in square brackets the overhanging values in the measurement unit of half pockets. [Left, Right]

The “0” shape number cannot be modified, its setting indicated that the tool has a normal size.

It is not possible to overwrite the already used shape number, which we have already assigned to tools. It is possible to edit such a shape number only in the way if no tool refers to it, i.e. if it is not the shape number of any tools.

First you have to search for a shape number, which contains the oversized of the tool to be taken up, or select a new, not-yet-used [0;0] shape number.

In case of a new, not-yet-used [0;0] shape number you have to write the oversize of the tool into the text fields “To the left” and “To the right”.

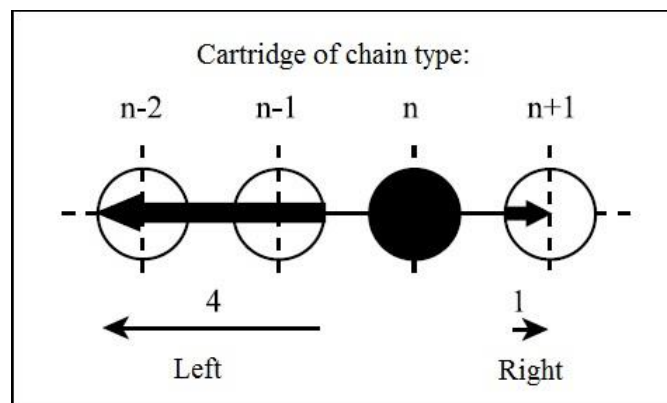
SingleToolEdit Channel1

Tool Row Id.: Cartridge: 10 Pot: 1		H index <input type="text" value="0"/>
ToolType Num. <input type="text" value="5"/>	Length geom. (mm) <input type="text" value="0"/>	Length wear (mm) <input type="text" value="0"/>
NickName <input type="text"/>	D index <input type="text" value="0"/>	
Tool info <input type="text" value="UENCI"/> <input type="button" value="Edit"/>	D geom. (mm) <input type="text" value="0"/>	D wear (mm) <input type="text" value="0"/>
Tool Fig. No. <input type="text" value="0 [0;0]"/>	Q value <input type="text" value="0"/>	
Left dir. <input type="text" value="0"/> Right dir. <input type="text" value="0"/>	Spindle <input type="text" value="0"/>	
Life Status <input type="text" value="Disabled"/>	Feed <input type="text" value="0"/>	
Life Cnt <input type="text" value="0"/>	<input type="button" value="✓"/> <input type="button" value="✗"/>	
Life Max <input type="text" value="0"/>		
Life Notice <input type="text" value="0"/>		

Interpretation of left and right directions:

To the left: decreasing pocket numbers

To the right: increasing pocket numbers



After this we have to click on the green tick at the bottom of the window and the modifications will be saved.

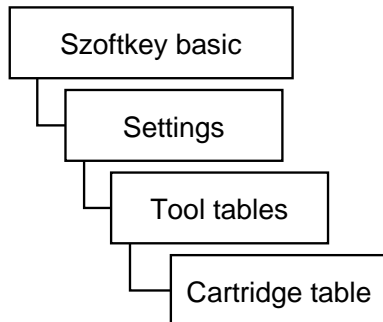


Attention! At certain machine constructions it is possible to set the characteristics of the tool only during the addition of the tool!

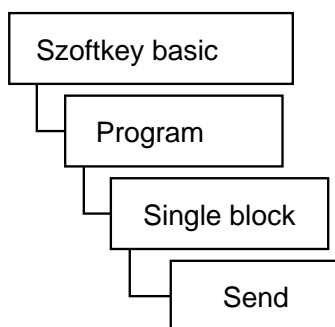
8.4. Deletion of a tool from the store

Note: The exact procedure of deleting a tool is determined by the machine constructor, the tool deletion presented here is only an example, as it is done typically in this way; however, your machine can be different from this. Please contact the machine constructor for more details about the procedure of deleting a tool.

Push the “T del” (T deletion) button on the machine control panel! The button will stuck in (its background starts to light).



For the deletion of a tool we have to know its exact location in the storage (magazine and pocket number). You can view the location of the tools in the tool store in the base status of Softkey, by pushing the “Settings/Tool tables/Tool place table”.



In the base state of the SOFTKEY select the menu option “Program/Unique block”!

Enter the “M62 XXXX YYYY” code where XXXX is the magazine number of the tool to be deleted, and YYYY is its pocket number. For example: if we would like to delete the tool located in Pocket 1 of Magazine 10, we have to enter the code “M62 0010 0001” without spaces.

Execute the unique block! (Push the “Send” button on the Softkey row, then the START button on the machine control panel.)

Usually the takeout of tools takes place through the main spindle, therefore if there is another tool in the main spindle, the control unit will first change it for the tool to be deleted.

After this, the control unit will get into the STOP status and in the form of a message it will ask whether to delete tool X or not? where at the place of X we can see the T code of the tool to be deleted. For example, during the deletion of tool T1000 we will see the following message:

00002800 **Do you want to delete tool T5? (Yes: Start/No: Reset)** 12/06/2012 09:39:30 AM MM760 Mill 09:40:31

SBLOCK STOP mm PLC

Single block

M62 T00010001

Row	Cartridge	Pot	ToolTable Row	Tool Fi...
1	1	1	T5 [6]	0
2	1	2		
3	1	3	T2 [2]	0
4	1	4	T1 [1]	0
5	1	5		
6	1	6		
7	1	7		
8	1	8		
9	1	9		
10	1	10		
11	1	11		
12	1	12		
13	1	13		
14	1	14		
15	1	15	T3 [3]	0
16	1	16		

Row	Cartridge	ToolTable Row	Tool Fi...
17	S Spindle [10]	T10 [4]	0

Win-s
PLC
PLC
PLC
MOP
Key
Auto
Space

Push the START button on the machine control panel! Then the control unit will request the takeout of the tool in a message.

00002600 **Remove T5 from spindle** 12/06/2012 09:44:04 AM MM760 Mill 09:44:08

SBLOCK STOP mm PLC





According to the design of the machine, take out the tool then push again the START button.

Switch off the "T del" (T deletion) button on the machine control panel!

9. Program Execution Start and Stop



You can start the program execution by using the **CYCLE START** button. The program execution can be stopped by the following buttons and functions:

- the **STOP**  button
- the **RESET**  button
- **SINGLE BLOCK**  button (this will cause the program to stop at the end of the block)
- **M00** Program Stop Command (this will also cause the program to stop at the end of the program)
- **M01** Optional Stop Command (the program will stop only if the **OPTIONAL STOP**  button is on)
- **M02** and **M30** commands will only be active at the end of the program.

9.1. Starting the Program Execution



The program execution will begin when you hit the **CYCLE START** button and chose one of the following modes:

- If a single block mode is selected, then choose **Jog**, **Incremental Jog** or **Manual Handle Feed Mode**.
- If a program is selected for execution from the program memory or from DNC, then choose **Automatic Mode**.
- **Manual Data Input Mode**

The **START** state is indicated by the START message in the 3rd status field window.

9.2. Cycle Stop



To cancel the **START** state, press the **STOP** button during the program execution and the axes will slow down and stop. The execution of **M**, **S**, **T**, and **B** functions will be continued until the end of block. The **STOP** state is indicated by the **STOP** message in the 3rd status field window.

The **STOP** button is ineffective in the following cases:

- **G63**, i.e. state of override inhibit.
- If certain values of **#3004 variable** are present
- During thread cutting **G33**, thread cycle **G76** and tapping cycles **G74** & **G84**.

9.3. Reset



If the **RESET** button is pressed during **program execution**, the **START** or **STOP** state will be canceled and the axes will decelerate and stop. In an operation where the feed override or the **STOP** button can't stop the cycle (i.e. **G33**, **G74**, **G84**), the **RESET** button can be used instead. Certain global messages are also cleared from the message field of the status bar when the **RESET** button is used. The **PLC** program stops the spindle rotation and suspends all operations which are currently under execution.

The **RESET** button has other effects in different modes as well.

Reset in Automatic Operation Mode

If the **RESET** button is pressed during the automatic operation mode, there won't be any **INTD** (interrupted) states, the program execution will stop fully, and the cursor will jump to the first line of the program. If you push the **START** button again, the program execution will start again and the control will be reset to its basic state, which is set by the parameters. The program execution can only be continued from a block that was interrupted using the **Block search** command. Please see chapter 11.6 "starting of automatic operation mode after block search".

Reset in Manual Data Input Mode

If the **RESET** button is pressed during the manual data input mode, the program execution is interrupted and it cannot be restarted from that interruption point, only from the very beginning.

Reset in Single Block Mode

If the **RESET** button is pressed during the single block mode, then the block is cleared from the buffer. If the same block needs to be executed again, the single block must be reselected and the **ENTER** key in the **SOFTKEY** line must be pressed again.

9.4. Programmed Stop M00

If the control reads the **M00** code during the program execution, the block that contains the **M00** command will be executed. The control will then go into the **STOP** state and the machining will stop.



The machining can be continued by pressing the **START** button.

9.5. Optional Stop M01

If the control reads the **M01** code during the program execution, the block that



contains the **M01** command will be executed. If the **OPT STOP** switch is on, the control will then go into the **STOP** state and the machining will stop.



The machining can be continued by pressing the **START** button.



If the **OPT STOP** switch is off, the **M01** command will be ignored and the machining will continue.

9.6. End of Program: M02, M30

The **M02** and **M30** codes indicate the end of the main program. If the control reads them, then the machining will stop and the **START** state will be canceled. The cursor will jump back to the beginning of the program listing and the program can be re-



started by pressing the **CYCLE START** button. If you don't use the **M02** or **M30** commands, then the % symbol will cause the same effect and the commands that belong to both codes will not be executed. I.e. **spindle stop** and **coolant off** will not be executed.

10. Program Execution Intervention

10.1. Conditional Block Skip

You can program a conditional block skip **/n** at the beginning of a block. The "n"



represents the block number. There is a **block skip switch** button on the operator's panel for activating the **/n** command. If the switch is **on**, then the block will be skipped. If it is **off**, then the block will be **executed**.

If you want the block skip command to be recognized in the proceeding block, set the **Program/Execution Config** parameter (**N1337**) CIBB bit to 0. In this case, the conditional block skip command suppresses the block buffering. If you use the **G41** or **G42** commands, the contour will distort. Due to the block buffering, it is enough to set the conditional block switch during the execution of the previous block.

If the **/n** command does not suppress the block buffering, set the **Program/Execution Config** parameter (**N1337**) CIBB bit to 1. In this case, if you use the **G41** or **G42** commands, then the contour will not distort but the conditional block switch must be set before program execution.



The **block skip switch** button can be selected in the software machine panel or in the NCT machine operator's panel. The control can handle another 8 conditional block switches, which can be used optionally.

10.2. Increasing the Feed rate Using Rapid Traverse Jog Button

If the axes are moving at the programmed feed rate during program execution, the



rapid traverse jog button can be pressed and the control will increase the feed rate. If the rapid traverse jog button is released, the feed will be reset to the programmed feed rate. The acceleration will be determined by the value set at **parameter N0313 Feed MULT**. The programmed feed value will be multiplied by the number set at this parameter. If the value of this parameter is 1, the feed rate remains unchanged even when pressing the **rapid traverse** button. If the value is set for 2, the feed will be doubled.

11. Debugging the Part Program

The following functions help the testing and debugging of the part program:

- Single Block Execution
- Dry Run
- Locking Possibilities
- Test Run
- Calculation of the Cutting Speed
- Description of the G and M Codes

These functions are not separate operation modes, but conditions influencing the program execution in the automatic mode.

11.1. Single Block Execution

Single Block Execution can be selected during the automatic and the Manual Data Input Mode. The control stops after the execution of each block. An exception is when thread cutting blocks (**G33**) follow each other. Another exception is when the suppression of a single block execution has been programmed by giving the macro variable **#3003=1**. The machining continues with the **CYCLE START** button.

11.2. Dry Run (all feed rates at high speed)



The **DRY RUN** button can be turned on in the software machine panel or in the NCT machine operator's panel. Once turned on, all feed movement is at a higher speed, according to the value set at the parameter. It executes every movement at the mm (or inch) / minute feed rate, independent of whether **G94** or **G95** is programmed.



Note: Generally the **DRY RUN** button can only be activated in one of the manual operation modes while the axes are stopped. Further details are available from the machine tool builder of each individual machine.



11.3. Machine Lock Function

If the switch is on, then the interpolator does not give out any command to the position control loop of any axis. Also, the PLC can not execute functions at all either.




If you use the NCT machine operator's panel, use the **Machine Lock** button to switch on this function. The locked state of the machine is indicated by the **M.LOCK** caption in the 8th field of the message panel.

The **machine lock** and **dry run** functions can be combined, meaning that both can be switched on at the same time. The **POSITION** screen and the **GRAPHIC POSITION** screen show the path already drawn. If the **dry run** switch is on, then the

feed is high. If it is off, then the feed is at the programmed feed rate value. If the **machine lock** is switched off, then the momentary axis positions can be seen again on the **POSITION** screen of the control unit.



Note: Generally the **MACHINE LOCK**  button can only be activated in one of the manual operation modes while the axes are stopped. Further details are available from the machine tool builder of each individual machine.

11.4. Function Lock



The **Function Lock** button does not allow the control to execute any functions sent to the PLC (i.e. **M, S, T, B**). To activate this function, press the button on the software machine panel or in the NCT machine operator's panel.

11.5. Axis Lock

This function allows you to pick out any single axis and lock it separately. This way, any other axes that are not locked down can still move freely. The different auxiliary functions and axes locking possibilities can be combined optionally. For example, it is possible to only lock a single axis while all other functions operate normally. Further details are available from the machine tool builder of each individual machine.

11.6. Test Run



To activate this function, press the **Test Run** button on the software machine panel or in the NCT machine operator's panel. The test state is indicated by the **TEST** caption in the 8th field of the status bar.


When the key is activated, the interpolator does not give out any command to the position control loop of any axis. Also, the PLC can not execute functions at all either.

When the control is in the **TEST** state, it can perform the syntactic and graphic test of the program. This way, the program execution (including the interpolation) is much faster than if the **MACH LOCK** and **DRY RUN** functions were being used simultaneously.



If the **TEST RUN** button is **OFF**, then the current axis positions can be seen again in the position display of the control panel.

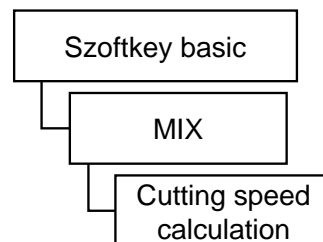


Note: The **TEST RUN**  button can only be activated in one of the manual operation modes while the axes are stopped. Further details are available from the machine tool builder of each individual machine.

11.7. Calculation of Cutting Speed

Velo calculator	
Tool diameter [mm]	Number of cutting edge
40	4
Cutting speed [m/min]	Cutting feed [mm/tooth]
160	0.1
Revolution [1/min]	Feed [mm/min]
1273.240	509.296

This window helps calculate different speeds and feedrates.



11.7.1. Constant cutting speed calculation (G96, G97)

The CNC control calculates the spindle speed, which is proportional to the specified surface speed at the position of the programmed coordinate value on the X axis.

Cutting Speed Calculation helps give a value for cutting speed during programming. Further details of the programming will be found in the description of the programming manual.

If the **Cutting Speed Calculation** window is active (the header is highlighted), the following buttons will be found in **SOFTKEY** line: **G94 - G95 - G96 - G97 - Copy - G20 - G21**

For calculating constant cutting speed, press the **G96** button. The descriptions of the different windows will change according to the selected calculations.

Velo calculator			
Tool diameter [mm]	Number of cutting edge		
40	4		
Cutting speed [m/min]	Cutting feed [mm/tooth]		
160	0.1		
Revolution [1/min]	Feed [mm/min]		
1273.240	509.296		
< G96	G94	Copy	
<	F1	F2	F3
<			

- Type in the values in the fields you selected and the result will be put in the lower window pane.

Feed selection by mm (inch) / minute (G94) and mm (inch) / revolution (G95)

G94: mm/minute or inch/minute or degree/minute feed

G95: mm/revolution or inch/revolution or degree/revolution feed

11.8. Description of G and M codes

SK > MIX > G codes or SK > MIX > M codes

The **G** and **M** codes are organized in groups. Within a group only one code can be activated. This window contains the list of the codes and shows the group division of **G** or **M** codes and shows which code is currently activated in the group (the activated one has a darker background).

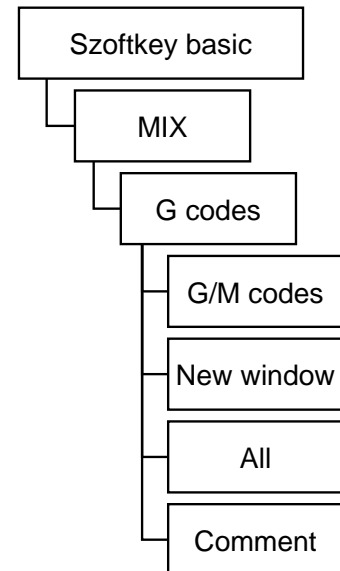
If any of the code windows are activated, the following buttons can be found in the **SOFTKEY** line:

M / G codes: With this button you can switch between the **G** and the **M code** windows. If you select the **G codes** button, then the window will change to **M codes** and vice versa.

New window: You can open another **G/M code** window by setting the old one for checking the **G codes** and the new one for **M codes**. This way you can see both windows at the same time.

All: If this button is switched on (darker background), both groups of codes will appear separately. If it switched off, then the codes will appear together without the group display.

Comment: If this button is switched on (darker background), a description of each code will appear.



12. Interrupting and Restarting the Automatic Operation

In many cases the automatic operation needs to be interrupted and restarted at a different time when the machining needs to be continued. There are several scenario cases for this, listed below:

- During machining, the program may need to be corrected if there is a syntactical error.
- In case of a tool breaking, then the broken tool must be retracted and replaced, before the machining can be continued with the a new tool (and possibly new offset values).
- During certain parts of the machining, if the results need to be checked and the machining needs to be continued later on.
- If there is an obstacle during the machining (for example, if a fixture is in the way of the tool path and it needs to be corrected).
- If the machining has to be paused in case of power outage and the machining needs to be continued from the interruption point.

The above cases require different types of intervention from the operator and they must be corrected by means of different control services, which are listed below.

12.1. Interrupting the Automatic Operation

During the automatic operation, the following interventions can cause an interrupted state:

- Emergency stop state (when pressing the **EMERGENCY STOP** button)
- Changing the operation mode

The interrupted state is indicated by the caption **INTD** in the 3rd field of the status bar. In the interrupted state, lots of manual intervention can be done on the control, which can be registered or canceled when the program will restart.

12.2. Restarting the Automatic Operation

The automatic operation can be restarted in three ways from the **INTD** state:

- Unconditionally
- The **BLOCK RESTART** condition
- The **BLOCK RETURN** condition

In the first case, if we moved the slide away from the point of the interruption, the slide will go to the end point of the block. If the program has increment commands, the slide will go to the very first absolute positioning point following the initial interruption, then it will go back to the original path.

In the second case, it goes to the start position of the interrupted block and it will re-run the whole block.

In the third case, it moves back to the interruption position and continues the machining from there.

12.3. Unconditional Restart the Automatic Operation

If the automatic operation is started unconditionally from an interrupted state (**INTD**), after the **CYCLE START** button is pressed, the control goes to the end position of the interrupted block and the machining will continue from there. The typical applications of this function are the following:

- The errors must be canceled in order for the machining to continue. In this case the error messages were coming from the program preprocessor module or from the **PLC**.
- During the machining, the cutting parameters might need some changes. It could be the change of the spindle revolution (**S**) or the feedrate (**F**) by means of manual data input. The machining can be continued afterward.
- In some simple cases, the end position or measure data needs to be modified without the part program modification.
- If there is an obstacle of the machining and it needs to be corrected (i.e. a fixture is in the way of the tool path).

12.3.1. Interrupting Simple Movements

1st case: Interruption of a linear interpolation parallel to an axis in a **G40** state.

Let us examine the sections of the following sample programs:

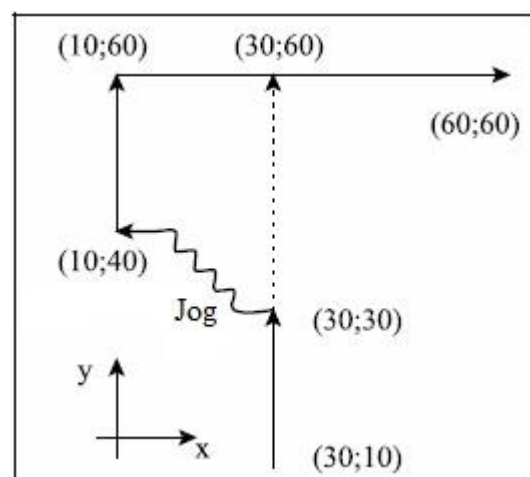
Program No 1:

```
...
N60 G90 G0 X30 Y10
N70 Y60
N80 X60
...
```

Program No 2:

```
...
N60 G90 G0 X30 Y10
N70 G91 Y50
N80 X30
...
```

The Programs No. 1 and 2 are moving on the same tool path. However the data specification of the 1st one is absolute, while the 2nd one is incremental. The interruption occurs in block N70. The movement was stopped at the position X=30, Y=30. The automatic mode was interrupted by the operator and the slides were moved to position X=10, Y=40 by manual operation. He returns to automatic mode afterwards and presses the **START** button. Then the movement programmed in block N70 will be finished. The Y Axis moves to the programmed position Y=60, regardless of how the program was written. The X Axis does not move in block N70, it only returns to the programmed original tool path in block N80, where the X movement was programmed. If there are several blocks without the X



axis programmed, the X axis will go back to the original path in the block, whichever contains the X command.

You can have the same movement when you switch over to single block mode at the beginning of the block N70 and move the slides away manually then return to the previous point. We can press **CYCLE START** afterward.

2nd case: Interruption of a linear interpolation not parallel to an axis in a **G40** state.

Let us examine the sections of the following sample programs:

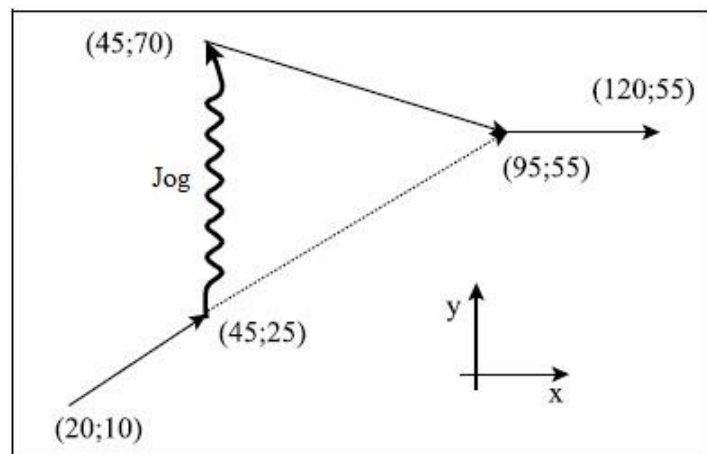
Program Nr. 1:

```
...  
N40 G90 G0 X20 Y10  
N50 X95 Y55  
N60 X120  
...
```

Program Nr 2:

```
...  
N40 G90 G0 X20 Y10  
N50 G91 X70 Y45  
N60 X25  
...
```

The Programs No. 1 and 2 are moving on the same tool path. However, the data specification of the 1st one is absolute, while the 2nd one is incremental. The interruption occurs in block N50. In position X=45, Y=25 the movement is stopped, the automatic mode is interrupted and the axes were moved to position X=45, Y=70 by means of manual operation.



Afterwards, we return to the automatic mode and press the **START** button. The movement that was programmed in block N50 will be finished. The X and Y-axis were moved along the oblique straight line from their momentary position to the programmed position X=95, Y=55. It doesn't matter if the tool path was written in absolute (program No. 1) or incremental programming (program No. 2). We can have the same movement in the following way when you switch over to single block mode at the beginning of the block N50 and move the slides away by manual operation, then return to the previous point. We can press **CYCLE START** afterward.

12.3.2. Interrupting the Drilling Cycles

Look at the following cycle:

```
...  
G17 G90 G81 X100 Y70 Z-60 R2 F200  
...
```

Due to the interruption, the canned cycle consists of 3 partial blocks:

1st partial block: Positioning in the selected plane. This block will be handled by the control as though it only had one or two programmed coordinates. (**G0 X100 Y70**)

2nd partial block: Positioning to the point R. This partial block is a positioning block along an axis. In this example, the Z-axis is moving to the position specified by the address R: **(G0 R2)**

3rd partial block: Hole machining and retraction to the initial position **(G98)** or to R-point **(G99)**. In this case, the end position of the partial block is the initial position of the R-point.

At the end of all three partial blocks, the control stops in the single block mode.

3rd case: Interruption of a drilling cycle during positioning in the selected plane.

The movement was interrupted while moving to the position X=100, Y=70 (as in the sample block). We used manual operation then returned to automatic mode and pressed the **START** button. The movement was similar to the 2nd case. The axes moved to position X100 Y70. It is possible to have a program where one of the axes was not programmed (e.g. Y) in the selected plane. We can move the slide in this direction as well as the Z direction. The return movement is similar to the 1st case, which means that none of the Y or Z-axes is moving when you press the **START** button.

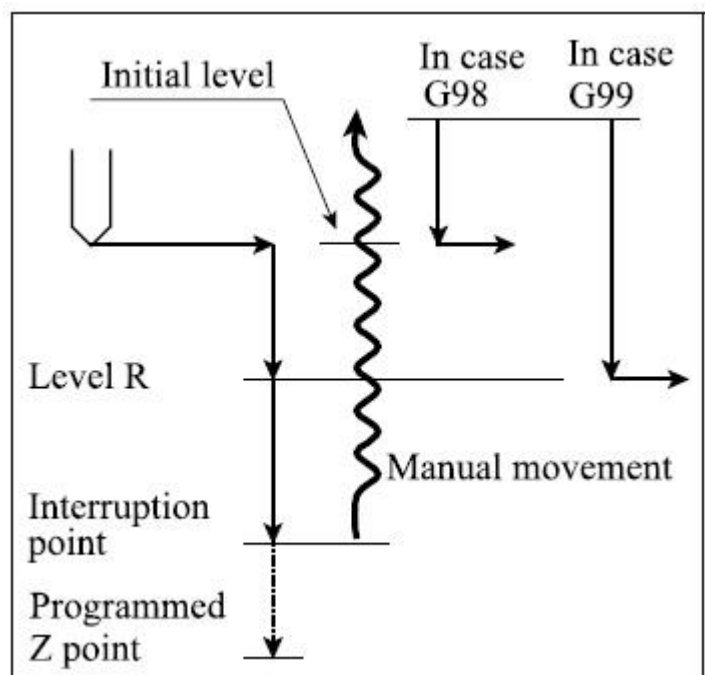
4th case: Interruption of the drilling cycle in the course of positioning to R.

G17 G90 G81 X100 Y70 Z-60 R2 F200

In the above example, the movement is interrupted while positioning in the Z direction to the R=2. Manual operation was used afterwards. Upon returning to the automatic mode and pressing the **START** button, the axis will move to the position in Z direction to R=2 similar to the 1st case. If we moved the axis in plane X-Y after returning to automatic mode and pressing the **START** button, then the X and Y will not make a return movement. The X and Y coordinates of the hole can be modified by the operator by means of manual intervention during the machining.

5th case: Interrupting the drilling cycle during drilling operation.

You can interrupt a drilling cycle while the machine is actually drilling as well. If you restart it with the **START** button, it will move to the initial level **(G98)** or to the level R **(G99)**. Afterwards, the machining will be continued with the execution of the forthcoming block. A repetition count can also be programmed in the block as well. If the cycle was not at the last hole drilling process after returning to initial or R-level, it will continue machining by positioning



above the next hole. There are two ways to interrupt the drilling operation:

1. While pulling back the drill, the operator notices that the depth of the hole is not deep enough. He can then press the **STOP** button, open any of the manual modes and execute the necessary manual operation. After returning to the automatic operation and pressing the **START** button, the control retracts the tool and continues the machining.

2. The operator can stop the drilling even before reaching the bottom of the hole and retract the tool in manual mode. He can return to the automatic operation and press the **START** button, which will cause the tool to move to the initial the R-level and the machining will continue from there. With this intervention, the depth of the hole can be corrected.

12.3.3. Interrupting the Machining in Tool Radius Compensation Mode

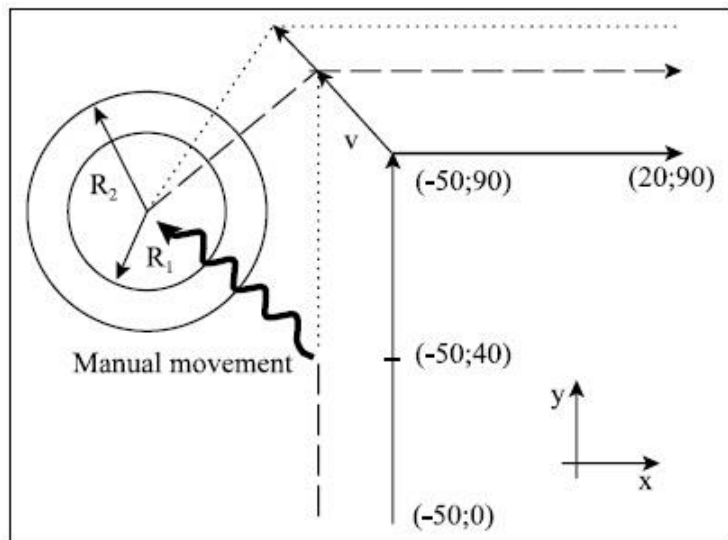
6th case: Interruption of a linear interpolation parallel to an axis in a **G41** or **G42** state.

Let us examine the part of the following sample programs:

Program Nr 1.
G41 G17 G90 G1 D1...
...
N80 Y90
N90 X20
...

Program Nr 2.
G41 G17 G91 G1 D1...
...
N80 Y90
N90 X70
...

Both programs are commanding the same tool path. The data specifications of the 1st one are absolute, while the 2nd one is incremental. If the machining is interrupted in position $X=-50$, $Y=40$, then the automatic mode is closed and manual operation is used. After returning to the automatic mode and pressing the **START** button, both axes will move in an offset mode to the programmed end position as seen on the diagram. The movements are the same in absolute and incremental mode



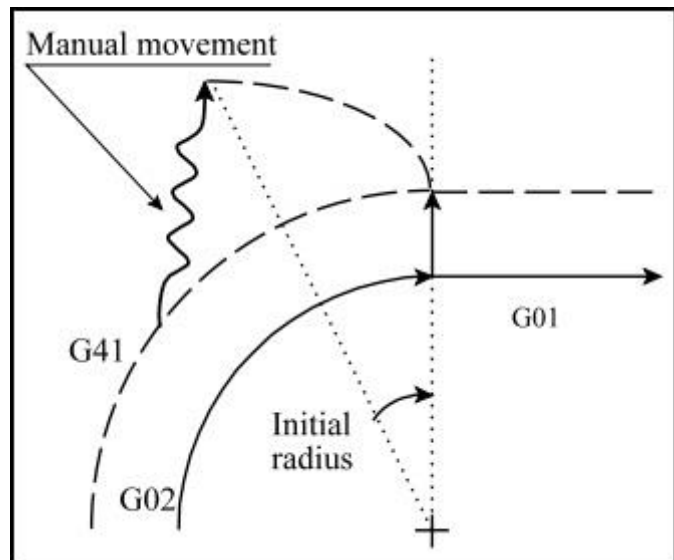
also. If there is a manual operation also made in the Z direction, there will be no Z movement in the automatic mode after you press the **START** button. This is the same as we discussed in the 1st case. In single block mode, the calculation and the return movement will also be the same as we discussed above. If the value of the tool radius has been changed, the machining will continue with the new value.

7th case: Interruption of a linear interpolation not parallel to an axis in a **G41** or **G42** state.

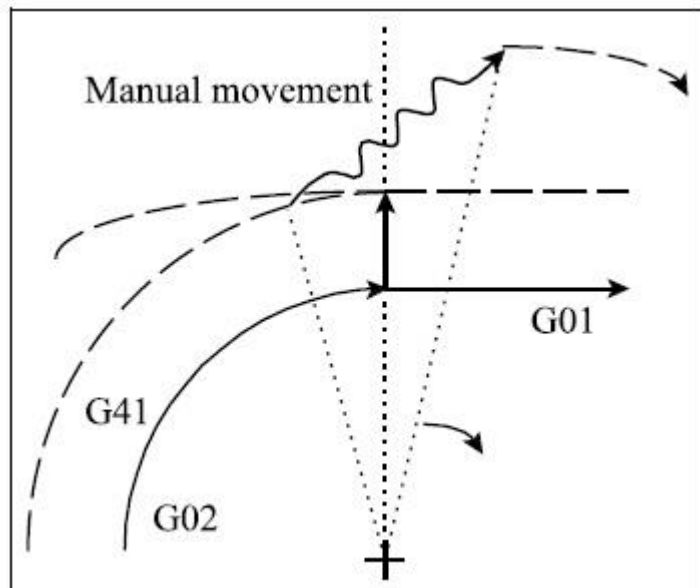
After returning to the automatic mode and pressing the **START** button, the tool path will be recalculated as we discussed in the 6th case.

8th case: Interruption of a circular interpolation in a **G41** or **G42** state.

If the circular movement is interrupted, then the tool will move away in manual mode. After the automatic mode is turned back on and the **START** button is pushed, the tool will return to the end position of the block where it was interrupted. The tool path will be a modified circle. The center of the circle corresponds to the center of the programmed one. Its starting radius will be equal to the distance between the center of the circle and the momentary tool position. The end point radius of the circle corresponds to the programmed one as well.




If we move the tool to a point, which is on the right side of the line between the end point and the center point of the circle, the control will do the same we described above. The radius of the return circle will be different, the starting angle will be close to 360° because the G02 direction.




12.4. Automatic Operation Restart with BLOCK RESTART Condition

Press the **BLOCK RESTART**  button and it will return to the automatic mode

and then push the **START**  button. Now the control will go to the start position of the interrupted block and continue the machining from there, meaning it re-executes the interrupted block. This function is typically for restarting the program after a tool breaks. After replacing the tool, the work shift and/or the tool compensation value can be changed also. When pressing the **START** button, it goes to the start position of the interrupted block with the new work and tool offset values.

12.4.1. Return to the start position of the block with manual operation


In the interrupted state (**INTD**) of automatic operation, the **BLOCK RESTART**  button can be switched on in any manual operation mode (**Jog**, **Incremental jog**, **Hand wheel**, or **handle feed**). After it is switched on, the return distance can be seen on the distance to go display for each axis. The axes can be moved continuously in any + or – direction to the position, until the distance to go will be 0. The axis movement will decelerate and stop automatically on the return position. Afterwards the axis cannot be moved any more from this position, only if the **BLOCK RESTART**



button is switched off. There is no need to completely move to the return position, the automatic mode and the **START** button can be turned on and the returning process will continue. With the help of this function, the return path to the start position can be made in the desired way, thereby avoiding the possible fixtures.

12.4.2. Return to the start position in Automatic Mode

Press the **BLOCK RESTART**  button and return to **automatic mode**

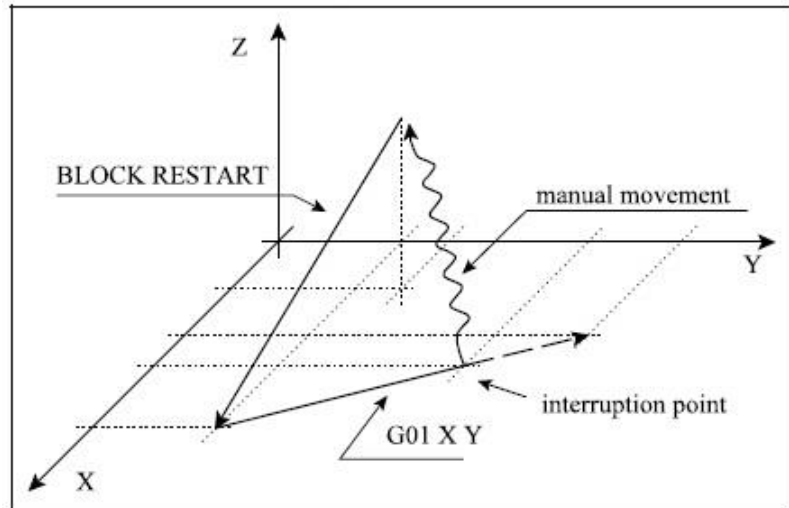
then push the **START**  button. The tool returns to the start position of the block by linear interpolation. If an obstacle is in the way of returning, the movement can be stopped with the **STOP** button. Afterwards, the return can also be continued by a manual mode.


12.4.3. Different return moves while the BLOCK RESTART is active.


The examples below are meant to show how to return to the start position of an interrupted block that has a movement command.

1st case: The control is in the **G40** state and the block is interrupted and restarts a linear interpolation.

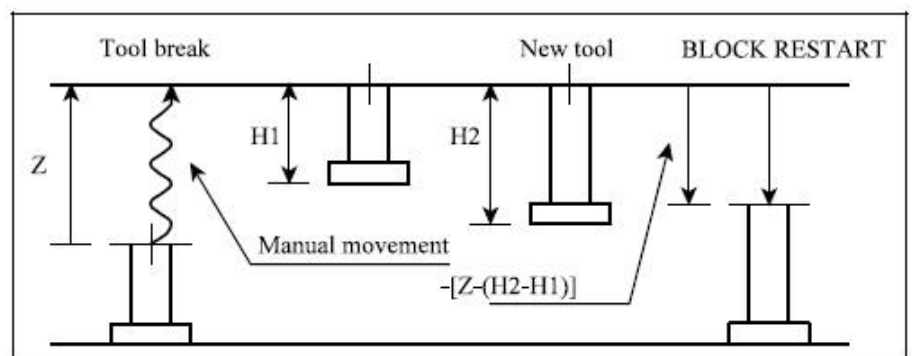
The graphic on the right shows how a linear interpolation in the X, Y plane is interrupted. This requires the manual mode to be set and a manual operation is used along all three axes. Then the automatic mode is turned back on and the



BLOCK RESTART  button is switched on. The **START** button is pressed and the control returns to the start position of the block.

The same exact thing will occur if the **BLOCK RESTART**  button is on and we use one of the manual modes. During the return process, all the axes are moving, even those which are not programmed in the interrupted block.

The return movement will add or subtract all the offset modifications that might be used. The graphic on the right shows how no movement was programmed in the Z direction in the interrupted block.



A similar scenario can occur when using a drilling cycle instead a linear interpolation.

Let us examine the following drilling cycle:

...

G0 X50 Y20 Z10
G17 G90 G81 X100 Y70 Z-60 R2 F200



If the drilling cycle is restarted with the **BLOCK RESTART** condition in effect, then there are three partial blocks in this cycle.

1st partial block: The tool is moving to a position in the selected plane. The start position in this partial block is: X=50, Y=20, Z=10.

2nd partial block: The tool is moving to the R point position. The start position in this partial block is: X=100, Y=70, Z=10.

3rd partial block: The tool is drilling and retracting to the starting position (**G98**) or to the R-point (**G99**). In this partial block, the start position is: X=100, Y=70, Z=2. At the end of all the three partial blocks, the control stops in the single block mode. If the drilling cycle is programmed with repetition count L, the above partial blocks are repeated.

2nd case: The drilling cycle is interrupted and it restarts while it is being positioned in the selected plane. In the drilling cycle listed above, the tool movement is interrupted while it is moving to the position of X=100, Y=70.

We moved to the start position of the block to the X=50, Y=20, Z=10 coordinate by



pressing **BLOCK RESTART** button. This movement is similar to the first case scenario.

3rd case: The drilling cycle is interrupted and it restarts while it is positioning to the R point. In the sample cycle above, the movement is interrupted while it is moving to the R=2 point along the Z axis. The tool is moved back to the X=100, Y=70, Z=10



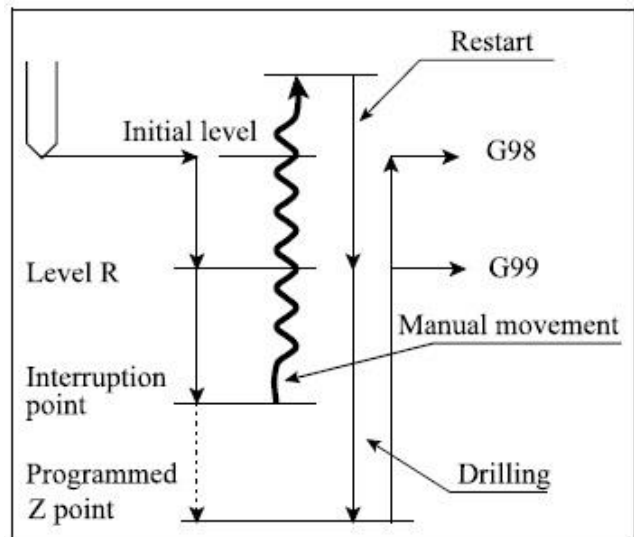
start position of the block once the **BLOCK RESTART** button is pressed. This is also similar to the first case scenario.

4th case: The drilling cycle is interrupted and it restarts during the drilling operation.

The tool is moved back to the X=100, Y=70, Z=2 starting position of the block when you press the **BLOCK**



RESTART button. If you press the **START** button, the tool will re-execute the drilling and the retraction.



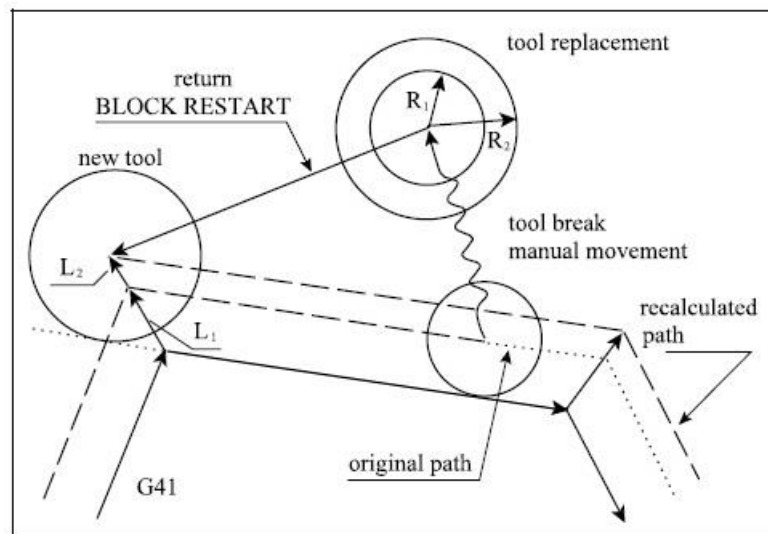
A similar scenario can occur when the cutter compensation is valid and the block is interrupted and it returns to its starting position.

In the cases below, the cutter compensation is on and all axes are in the selected plane. If the movements are out of the selected plane, the cases which are described in the **G40** state can be applied.

5th case: The tool returns to the start position of the block after the machining was interrupted in offset mode and the tool was at an outside corner. If you want to start the tool return, press the **BLOCK RESTART**



button. In this condition, the tool will start from the interrupted block. There will be the following calculation: the length of the offset vector will be multiplied by the length of the new corner radius of the tool, then the result will be divided by the length of the old corner radius.



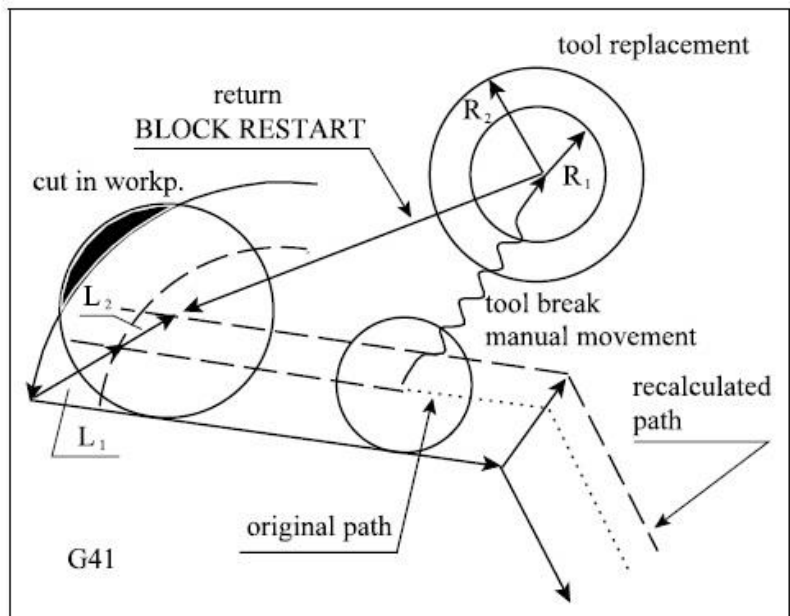
$$L_2 = L_1 \cdot \frac{R_2}{R_1}$$

Afterwards, the machining is continued on the tool path with the new offset value. During the start point return, all the axes will return to the starting point. The tool will move also in the Z direction if any manual movement has been made in this direction. If the tool length was also modified after the tool was replaced, the new registered tool offset will be taken into account during the return movement (see the 1st case).

6th case: Return to the start position of the block after the machining was interrupted in offset mode and the tool was at an inside corner. When the **BLOCK RESTART**



button is pressed, the machining of the inside corner corresponds to the case of the machining of the outside corner. As the diagram on the right side above shows, if one of the curves is a circle, then the tool will cut into the material.



This can be avoided if the tool is not returned fully to the start position, but stopped



earlier instead and the **BLOCK RESTART** button is switched off.

12.5. Automatic Mode Restart with BLOCK RETURN turned on

If the automatic operation was interrupted, then the manual operation will move the tool away from the work piece. The tool return movement to the interruption point can



be made while the **BLOCK RETURN** button is pressed. This button can be selected on the machine operator panel or on the software operator panel.

The control calculates the return position by including the possible changes of the work offset and the tool offset values. In the position display area, the difference between the return position and the present position will be shown. Afterwards, the return can be made by means of manual operation or automatically in automatic mode.

12.5.1. Returning to the interruption point by manual movement

Under construction!

In the interrupted (PENDING) state of the automatic mode, the BLOCK BACK switch can be switched on in any of the manual movement modes (MOVEMENT, STEPPING, MANUAL



WHEEL). After switching on the switch, on the REMAINING way display the distance necessary for the repositioning per axis will be displayed. The axes can be moved continuously in any (+/-) direction till the point at which the value of the remaining way becomes 0. The moved axis will automatically slow down and stop at the point of return. After this, we will not be able to move this axis from this position, only if we switch off the BLOCK BACK condition.

It is not necessary to fully reposition to the interruption point, but we can continue with the



returning by switching to automatic mode, upon START. By this function we can safely return to the interruption point on the desired path, by avoiding any potential obstacles.

12.5.2. Returning to the interruption point in automatic mode



In the automatic mode, in the switched on state of the BLOCK BACK condition, upon



the START by moving all axes at the same time along a straight line it will reposition back to the interruption point. If we detect an obstacle in the way of the returning, we can



interrupt the movement by the STOP button. After this we can continue with returning into any manual mode manually, too.

12.5.3. Cases of return under the BLOCK BACK condition



The cases of return under the BLOCK BACK condition are the same with those



described in case of the BLOCK AGAIN condition, with the exception that it will not go back to the starting point of the block but to the interruption point.

Returning to the interruption point after the interruption of a simple movement block

Case 1: (Under construction!): Interruption and restart of the straight interpolation in G40 state

The picture shows the case where we interrupt a straight interpolation in the X, Y plains, step out into some of the manual modes and move along all three axes. If, after this we return into automatic mode, and after switching on the

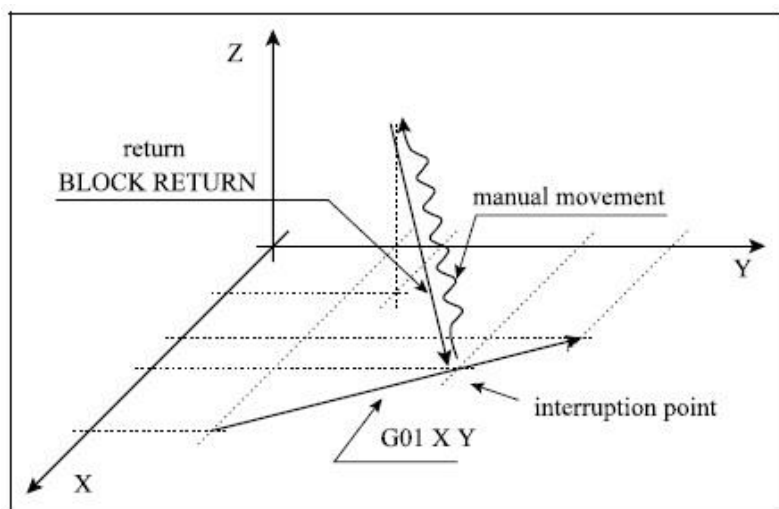


BLOCK BACK condition we push a START, or move in any manual modes in the switched on state of the

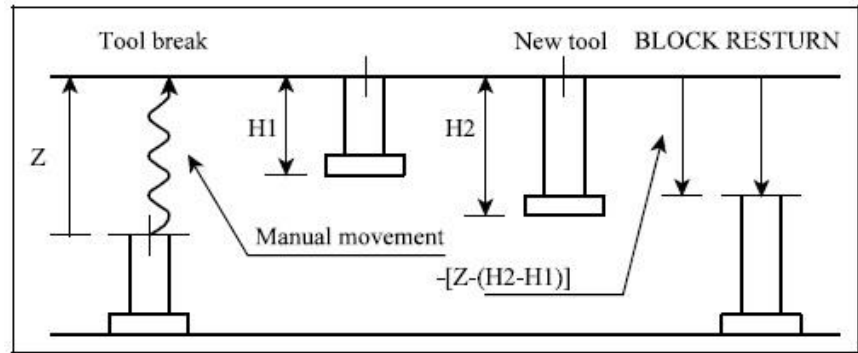


BLOCK BACK switch the control unit will return to the interruption point.

All - that means axes not programmed in the interrupted block - axes participate in the returning process.



If we modified a correction on any of the axes, such as along axis Z in our example, or if we modified the zero-point offset, it will go back to the position Z valid at the time of interruption by taking into consideration the new correction or zero-point offset.



Returning to the starting point of the block after the interruption of drilling cycles

Let's take a look on the following cycle:

...

G17 G90 G81 X100 Y70 Z-60 R2 F200

...

The drilling cycle restarted by the BLOCK BACK condition consists of 3 partial blocks:



Partial block 1: positioning in the selected plain. (G0 X100 Y70)

Partial block 2: positioning to point R. (G0 Z2)

Partial block 2: drilling and pull-back to the starting point (G98) or to point R (G99).

At the end of all three partial blocks the control unit will stop in a per block mode. If the drilling cycle is programmed by the L

repetition number, the above partial blocks will be repeated.

Case 2: Interruption of the drilling cycle and returning during positioning in the appointed plain

If in the cycle according to the example we interrupt the movement during the movement to the point in the position of X=100, Y=70, and we go back to the interruption point by the



BLOCK BACK condition, and movement can be originated back from case 1, i.e. all the axes will go back to the position valid at the time of interruption.

Case 3: Interruption of the drilling cycle and returning during positioning to point R

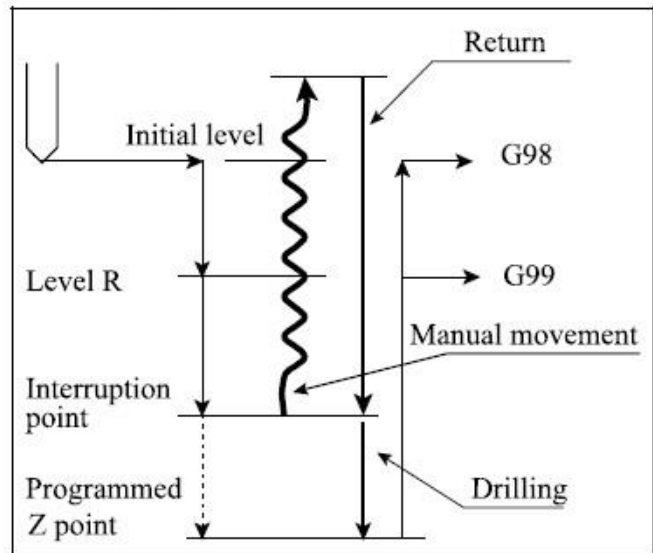
If in the above example, we interrupt the cycle during positioning to the Z-direction R=2



coordinate, and we go back to the interruption point by the BLOCK BACK condition, the movement can be originated back from case 1, too.

Case 4: Interruption and restart of the drilling cycle during a drilling operation

If we interrupted the drilling cycle during the drilling operation, and we restart the drilling operation by the BLOCK BACK condition, the movement can be originated back from case 1, too.



Returning to the interruption point after the interruption of contour following

The below indicated cases refer to axes being in the switched-on status of contour following, in the selected plain.

For the interruption of axis movements outside of the selected plain and for returning, cases described in G40 can be applied.

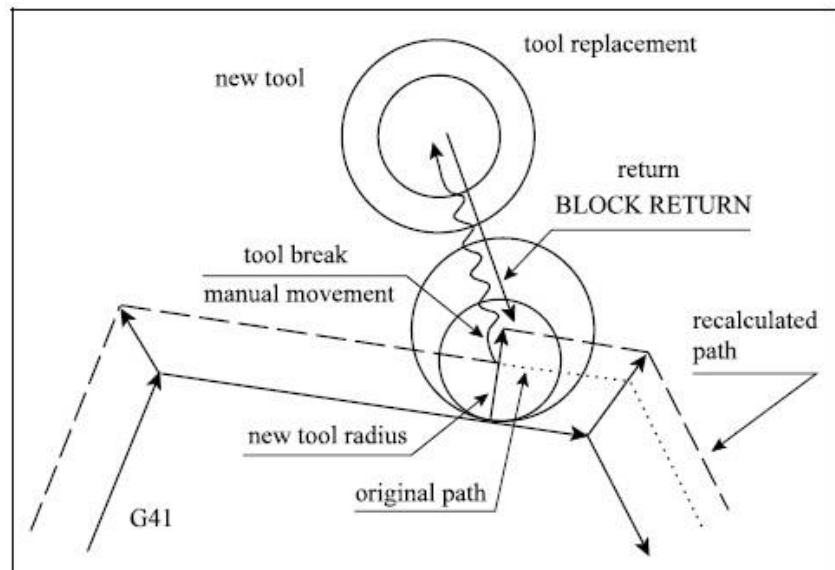
Case 5: Going back to the interruption point during contour monitoring, at the bypassing of external corners

Upon the BLOCK BACK



condition, it will set a vector rectangular to the interruption point, the length of which will be equal to the length of the new tool radius (provided that we changed the radius correction). After this, the machining will continue on the course recalculated for the new correction value, from the interruption point. During going back to the

interruption point, the control unit will reposition also on the other axes. Thus e.g. also in the Z direction, if the movement took place in this direction. If after the tool change, the length correction became modified, too and the length correction was recorded in direction Z, during the returning in Z direction it will take into account the new length correction (see the description of case 1.).

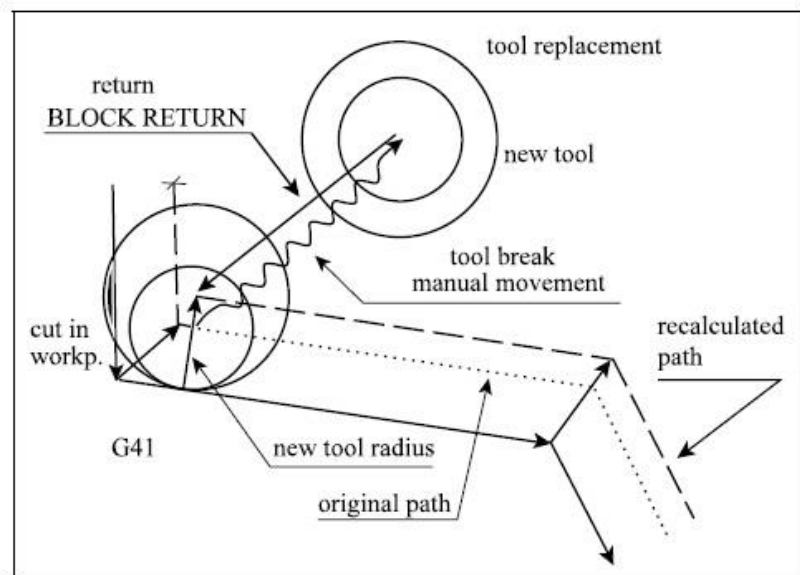



Case 6: Going back to the interruption point during contour monitoring, at the bypassing of internal corners

During bypassing internal corners, in the switched on state of the BLOCK BACK



condition, it will act in the same way as in case of external corners. However, in this case, as it can be seen in the picture, if the interruption took place near the corner and the radius of the new tool is bigger than that of the old tool, it may cut into the material. We can solve this problem by not navigating the tool back till the interruption point but first we



have to stop and switch off the BLOCK BACK  condition. After this we can operate according to the content of the chapter “Unconditional restart of the automatic mode”. The other option is to start a block search for the interrupted block; however, the description of this does not constitute the subject of this chapter.

12.6. Automatic Operation Start after Block Search

In the course of machining, there may be a need for executing the program selected for automatic mode not from the first block, but from a block within the program.

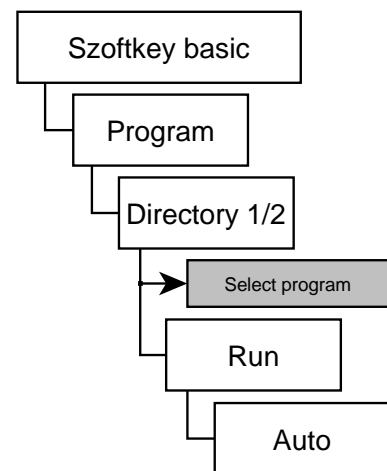
The block search can be activated on the following conditions:

- Automatic operation mode is on, but not in a **CYCLE START** state.
- There should not be an **INTD** state, meaning that the program execution must not be interrupted.
- You have to point at the desired block in the program.

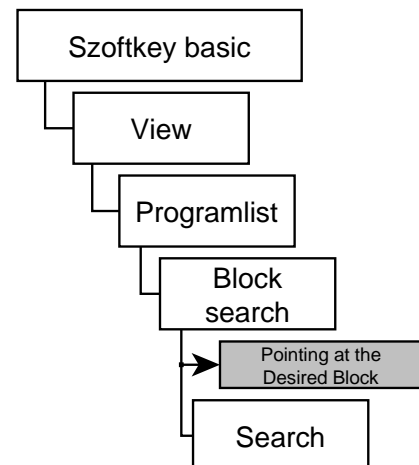
12.6.1. Pointing at the Desired Block

Pointing to the desired block of the program can be done in the following ways:

- 1) Select the program for automatic running.
Give the access path of the program in the directory, which is accessed after this operation:
SK > Program > Directory 1. Select the program, then press **RUN** in the **SOFTKEY** line.
(If it is a multi channel machine, you have to select a channel inside the **RUN** button).



- 2) List the program
Select: **SK > View > Program list**.



- 3) Search for the desired block in the program.
Press **Block search** in **SOFTKEY** line.
A text field will appear in the bottom of program list.



Type in the block number that you are planning to run through the program. In case of several results, you can skip between them by using the **>>** button next to the text field.

Press the **Row number** button on the **SOFTKEY** line and you can search by line number of the program.

Press the **Case sensitive** button on the **SOFTKEY** line and there will be a distinction between upper case and lower case letters.

The **Next** button is the same as the **>>** button next to the text field.

You can skip backwards as well during the search if you press the **Last** button on the **SOFTKEY** line.

- 4) You can also use the scroll bar of the program list window to select the desired block in the program.

12.6.2. Pointing at the Desired Block in a sub-program

If you have to point at a block in a sub-program, first double click the place where the sub-program is called and it will be listed. You can click again on any other block in the sub-program and it will be closed and the main program will be listed again. Do not click on a calling command in the sub-program because it will call another (nested) sub-program.

12.6.3. The SEARCH Command

In order to use the **SEARCH** command, you need to be in **automatic mode** and the running program should be displayed on the program list window. Select the desired program block through the following command chain:

SK > View > Program list > Block search > Search

The control will collect all the information in a table below the program list, which will represent all of the commands made since the beginning of the program. The message **SEARCH** will be displayed in the status field, showing that a block has been searched.

Let us examine a fragment of the following part program:

```
N010 G54 G15 G17 G40 G49 G80 G90 T1
N020 G0 X150 Y0 B90 M6
N030 T2
N040 G43 Z-5 H1 S500 M11 M3 M8
N050 G1 X-100 F200
N060 Y50
N070 X100
N080 Y100
N090 X-100
N100 G0 Z5 M5 M9
N110 X150 Y0 M6
N120 T3
N130 G43 Z-10 H2 S1000 M12
M3 M8
N140 G1 G42 X100 F300
N150 Y100
N160 X0
...
```

Start the block search for the **N150** block. In this case, the control collects the information you see in the green-tinted window.

The selected tool number:
T2.

The **M6** command is programmed in the **N110** block, which made the tool change specified in the **N030** block. Therefore, upon pressing the **START** button, the **T2** tool will be searched in the tool magazine.

The screenshot shows a window titled 'Program list > : \USB Disk\MONDAT KERES.prg'. It contains a list of program blocks with red arrow icons to the left. The block N150 Y100 is highlighted in green. Below the list, there are four sections with green-tinted boxes containing the collected information:

- Tool:** T2, M6, T3
- M-groups:** M8
- Spindle S:** S1000, M3, M12
- Positions:** X100.000, Y0.000, Z-10.000, C-11880....

At the bottom, there is a status bar showing '00:00:00.000' and 'Level 0 | L1 | Row 15/16'.

Tool change: M6

After preparing the **T2** tool, the tool will change because the **M6** command is programmed in the **N100** block.

Tool to be Prepared: T3

The **T3** tool is programmed in the **N120** block. In this case, the search for the **T3** can be done while the **T2** is still working. The above example illustrates how the data referring to the tool change is gathered. The tool change is done by means of the **M6** command in the machine. If the tool change is done simply with the function T, it will only gather the last code in the block. The method of gathering data referring to tool change can be specified in the parameter **1338 M06 (Program/Block Nr Search)**. If the value of this parameter is 1, it executes the data gathering according to the **M06** function.



Attention! You have the following program section:

T1

M6 T2

When using the **M6** command in the 2nd block, the tool will be changed. If you wish to program the tool preparation only, then program the **T code** in a block, which has no **M6** command. The correct version for this is shown below:

T1 M6

or

T1

T2

M6

T2

Spindle Gear Range Code: M12

In the **N150** block, the gear range **M12** code is valid because it was programmed earlier in the **N130** block. This code will be active.

Spindle Revolution Code: S1000

In the **N150** block, the spindle speed **S1000** command is valid because it was programmed in the **N130** block. This code will be active.

Spindle Rotation Code: M3

This command refers to the last spindle rotation code that is gathered when the **M3 command** is programmed in the **N130** block.

Coolant On Code: M8

This command refers to the last spindle rotation code that is gathered when the **M8 command** is programmed in the **N130** block.

Other Functions: M50, M51

Other **M** functions are listed in a separate line. In the sample program, the table has been rotated to the **B90** position, which was programmed in the **N010** block. For the table rotation, the **O9031** macro is called. The following program section can be found in the macro:

```
%O9031(TABLE ROTATION)
...
M50
G0 B#196
M51
...
M99
%
```

Since the table has Hirth toothed clamping, it must be unclamped before rotation by using the **M50** function. It must also be clamped after the rotation by using the **M51** function. In the course of a block search the control also reads the macros and gathers the information needed from there.



Note: In the **NCT 200** control family, even the user can define **M-code** groups. If there is a group which includes the **M50** and the **M51** commands, the search will be made in the following way: the last M code will be searched (which is in this group) and was programmed in the last block, prior the search. For details about M code group definition please refer to the description of the parameter.

Positions

The gathered positions are absolute values and the slides will go to those positions after the **CYCLE START** button is pushed.

The **N130** block is a milling operation of **G1** linear movement parallel to the Y axis on the **X100** coordinate from Y0 to Y100. This block is pointed at during the course of the block search, meaning that this will be the first block to be executed by control after the block search. Therefore, before executing N130 block, the control goes to the position of the **X100, Y0, Z-10** coordinate. The situation is the same in case of other interpolation (**G2, G3, G33**) movements, meaning that the table shows the starting position of the selected block. If the selected block contains rapid traverse positioning (**G0, block**), the positions of the table will show the end position of the selected block. If a drilling cycle is selected, the same positions will point to the end position in the selected plane.

Afterwards, the operator has to decide the proper order so that the control unit can reach the necessary conditions.

START Without Intervention

If the operator presses the **START** button without any manual operation, the control will reach the necessary conditions in the following order:

– First, it executes the selected functions according to their order in the table (i.e. in this case: **T2, M6, T3, M12, S1000, M3, M8, M50, M51**).

- Second, it executes the positioning on the axes which have a custom macro command appointed to them. In this case, it will move on axis B because it will generate a macro call once the move is finished.
- Finally, the tool positions itself along all axes at the same time. This positioning will be a rapid traverse movement and the rapid speed will stop 1000 increments (measured along the vectors) before the end position. The rest of the distance will be made by the programmed feed rate.

Manual Operation Intervention Before **START**

If a manual operation mode (jog, incremental jog, or manual handle feed) is selected before pressing the **START** button, then the **distance** / **distance to go** can be moved manually. It is similar to how we've described the **BLOCK RESTART** and the **BLOCK RETURN** conditions. There is no need to fully return to the searched position because the process can be continued by switching over to the automatic mode and pressing the **START** button instead. Afterwards, the order of the execution of the function corresponds to the methods discussed previously. The position calculated in the course of a block search is done in a way that will avoid hitting any possible obstacles.



Note: By stepping into another operation mode (like: **MDI**), the **BLOCK SEARCH** state will be canceled by the control and you will have to repeat the search again.

Other Intervention Possibilities Before START

It is possible to change the order of execution, and cancel the execution of other M - codes before starting the execution of data gathered in automatic mode. Touch one of the functions or coordinates (which should be taken out of the execution after the first **START** button is pressed) and the display's background will turn from green to yellow. If you touch this field several times, the colors will alternate in cycle.

When the **START** button is pressed, the yellow commands are not executed, only those which are not taken out of the execution (green background). After the selected commands (the green ones) are executed, the control assumes the **STOP** state and displays the commands in green, which are not executed in the first cycle. The above selection can be repeated several times.

Some of the other gathered M functions can be canceled. In this case, double click the function to be cleared and it will turn red. Press the **START** button and it will not be not executed, nor will it get cancelled either. Next time it will appear in red again. By touching them once, they will turn green, so next time they can be executed. The process begins again if you press the **START** button again.



Note: Positioning, tool changes and spindle commands cannot be canceled.

This process can be continued until all the gathered commands are executed. Then even the red commands, which are assigned to be cancelled, will disappear and the execution will continue according the usual automatic mode.

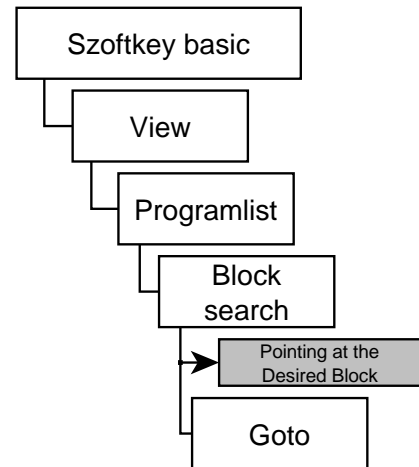
In the above example on page 82, the **M3** command can be taken out from the first execution cycle, as well as the movement Z-10. This way, we can prevent the tool from crashing during positioning. After pressing the **START** button, all functions except the spindle rpm start and the Z positioning, are executed. If the **START** button is pressed again, the spindle rpm will start and the tool will move to the work piece in Z-direction.

Two other functions were gathered: **M50** table unclamp and **M51** table clamp. As we mentioned before, the control also reads the macro **O9031**. There is the B movement command in there. The table unclamps and clamp commands are necessary to be carried out, which are commanded inside the macro. Obviously in this case the execution of the M50 and M51 commands is useless, therefore it can be canceled.

It is always the operator's responsibility to decide which functions are worth saving and which ones can be erased.

12.6.4. The GOTO Command

If you point to the desired block and press the **GOTO** key, the control will execute the program from the selected block after you press the **START** button. Contrary to the **SEARCH** command, it does not gather functions and positions from the beginning of program. It is the operator's duty to place the machine in the appropriate state and position. The effect of the **GOTO** command is the same as if we would command a **GOTOn** (n is the block number) at the beginning of the program.



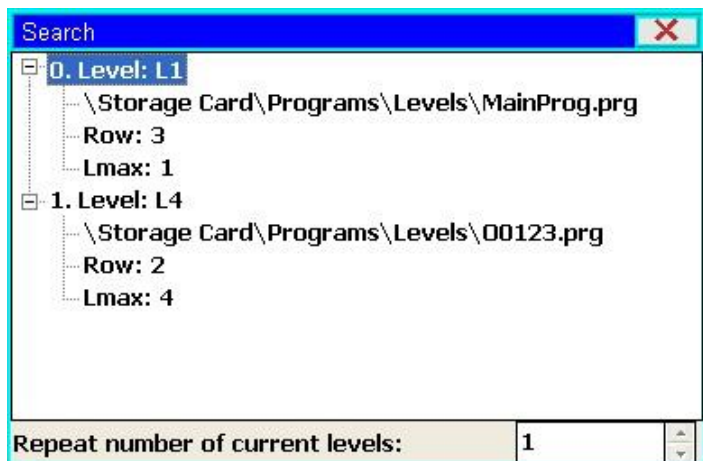
12.6.5. Giving a Repetition Number

If the program has a subprogram call and you wish to search for a block in the subprogram, it is possible to tell how many times the subprogram can run when you press the **START** button. When you use a **SEARCH** or **GOTO** command for a block in the subprogram, a window will show up to display the multi level sub-program call.

Next to all levels, you can find an **Lx** expression, where the **x** is the number of the repetitions. In each level, you will find the access path to the sub-programs and its name.

Next to the **Row** label, you can find the number where the subprogram was called. On the lowest level, you can see the number of the block you wish to find. Next to the **Lmax** label, you

can see how many times the sub-program would run normally. You cannot call the sub-program more times than this number when you use the block search.



In order to set the repetition for a sub-program, select a level then set how many times the sub-program shall be called, using either the **L++** or the **L-** buttons. You can find the number of the current level in a window at the bottom of the display. Here, you can also set how many more times the sub-program can be repeated by using the number scale. You cannot assign a similar repetition number like that to the main program.

In the **SOFTKEY** line, the **SEARCH/GOTO** buttons can be found again. After giving the repetition number, you can finalize its settings. The remaining tasks are the same as those described in the **command GOTO** and **command SEARCH** chapters.

13. Settings

13.1. Program settings

In order to modify the program settings, use the following command chain.

SK > Settings > Prog. settings

The **Prog. setting – NCT user** window will pop up.

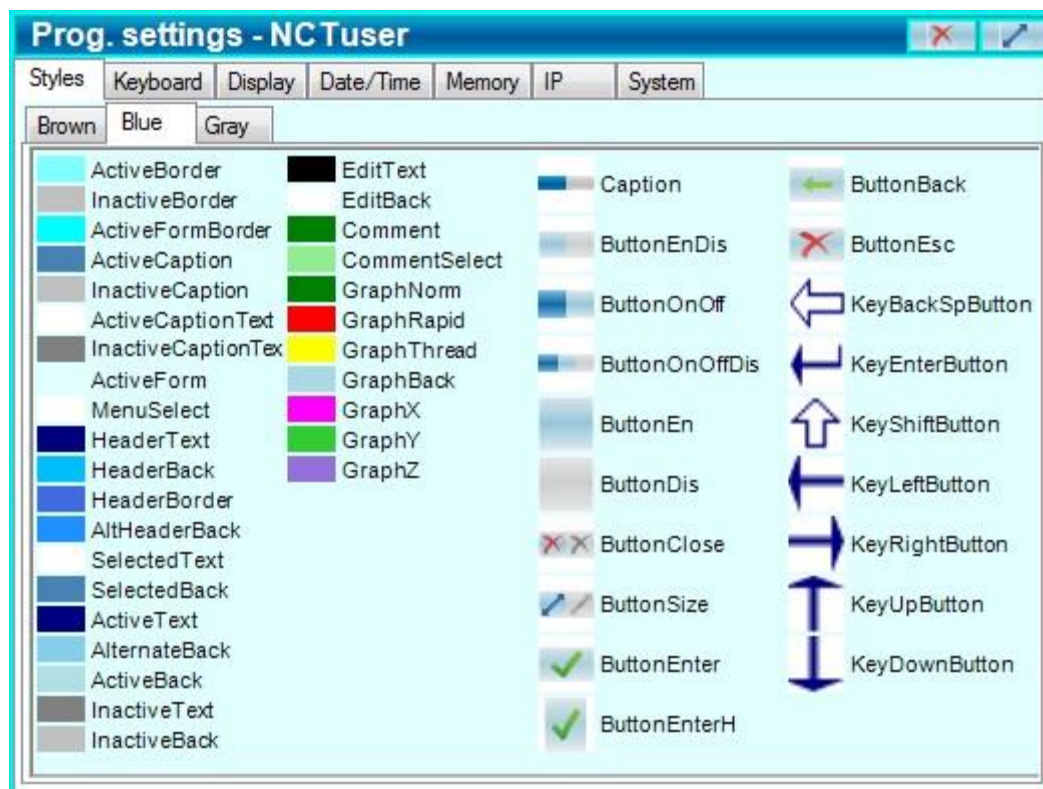
Here, you can set the colors, resolutions, language, network, time and other functions of the control panel.

In the header of the window, you can find following tabs:

- **Styles:** Setting up the styles of the control panel.
- **Keyboard:** Setting of the USB keyboard and the machine operator panel.
- **Display:** Setting of the display resolution and screen visualization.
- **Date/Time:** Setting the date and time, as well as registry settings.
- **Memory:** Displays the current amount of free and used memory.
- **IP:** Allows you to modify the network setting.
- **System:** Determines the frequency of various system backups.

13.1.1. Style adjustment

SK > Settings > Prog. settings > Styles



You have a choice between three basic colors: Brown, Blue and Gray.

Select the adequate color tab. On the right side of this window, you can see the different buttons, text background colors, sign layouts and pictograms in the style that you have chosen.

Save changes: You do not have to press any button for savings, these settings will become valid immediately.

13.1.2. The Sound of Error Messages

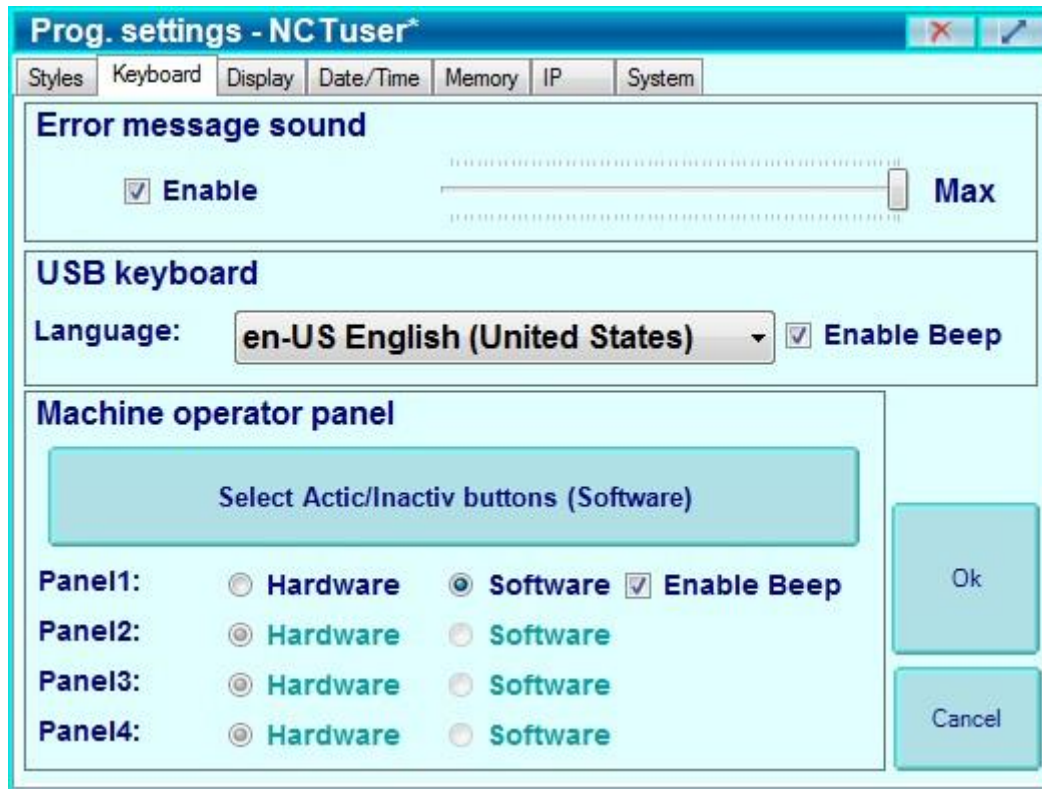
SK > Settings > Prog.setting > Keyboard

You can see three setting fields separated by the frames.



In the top **Error message sound** field, you can enable or disable the sounds of the error messages. You can set the sound volume by using the volume control slide.

13.1.3. The Language of the USB keyboard



In the middle **USB keyboard** field, you can set the language for a keyboard connected to the USB connector. You can enable or disable the beep of the keyboard click by checking the **Enable Beep** window.

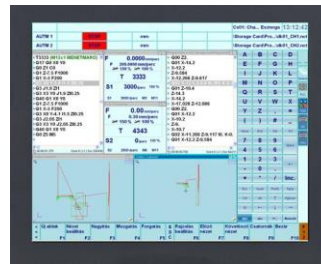
13.1.4. Which keyboard is active?

The bottom **Machine operator panel** field is used to select the active keyboard. Only a single keyboard can be activated at one time: either the physical or the software. If you have the first configuration, like the one on the left side, you can select whether you wish to use a physical or a software machine operator panel. The second configuration only allows you to use the software operator's panel.

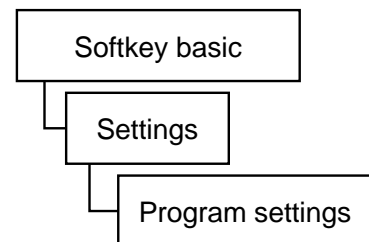
1. Configuration:



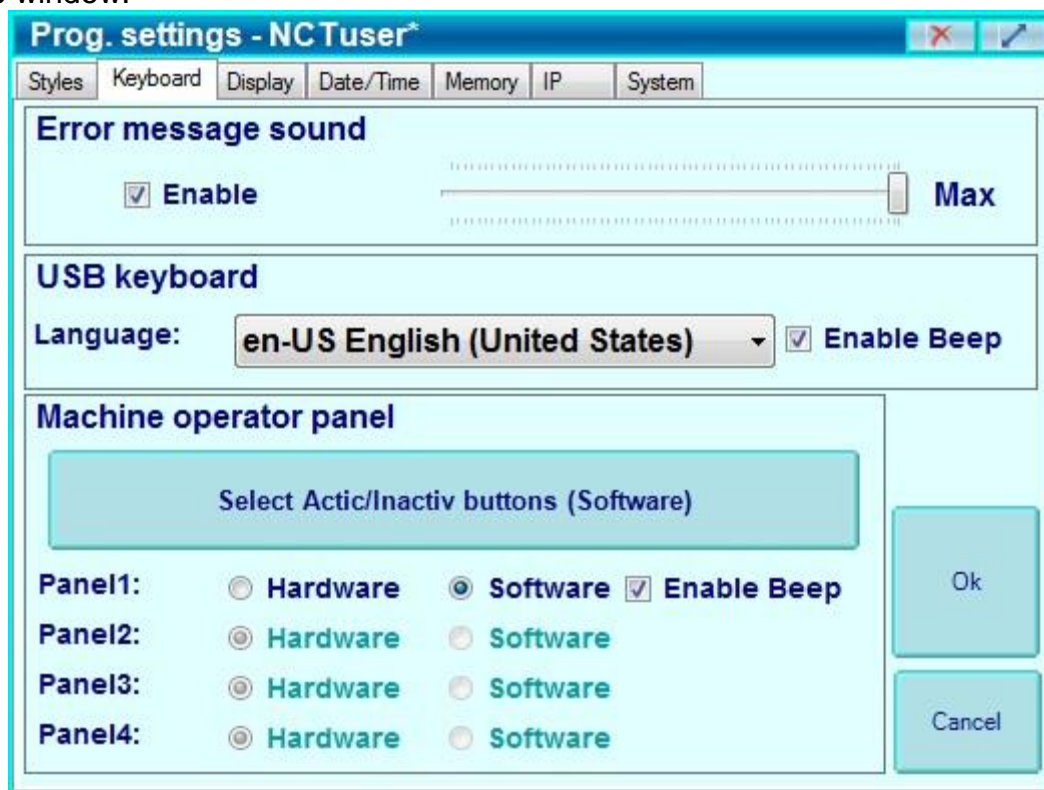
2. Configuration:



Menu map:



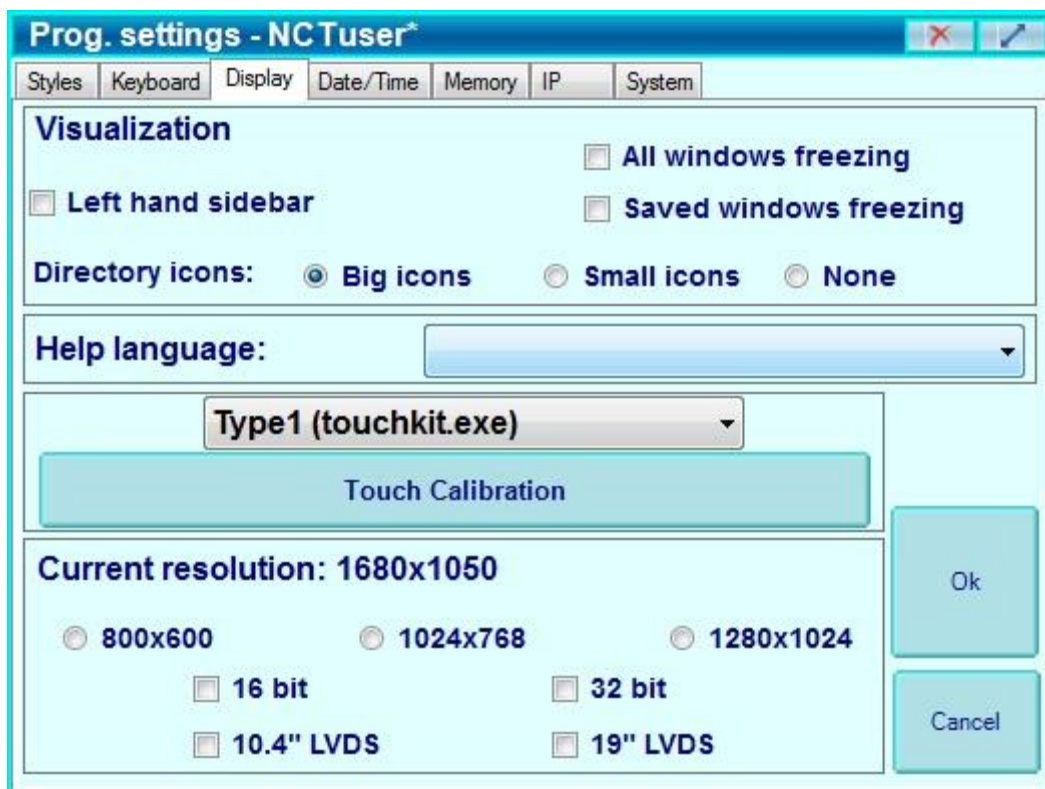
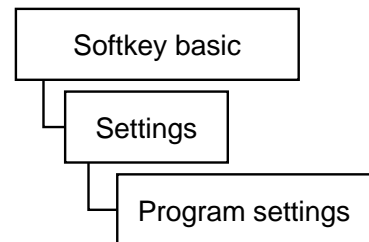
You can enable or disable the beep of the keyboard click by checking the **Enable Beep** window.



Save changes: To validate the changes you have made, press the **OK** button in the right corner of the window.

13.1.5. Freezing the windows

The **Display** settings allow you to determine the window size and the location of multiple windows that are visible on the monitor panel. If you want to freeze all of the windows, check the **All windows freezing** box in the Visualization section of the **Display** tab. When a window is frozen, the size button in the upper right hand corner of the window will turn grey or disappear. In that case, none of the windows can be moved or resized. Once it is frozen, you can double click on the header of the window to maximize it and fill up the full screen of control monitor.



Freezing can be done in two ways:

- You can freeze all windows by clicking the **All windows freezing** box.
- If you wish to freeze only a single window out of group, then check the **Saved windows freezing** box.

If you need to save a list of frozen windows, do the following:

Open the window you wish to freeze and set them to the adequate size and place.



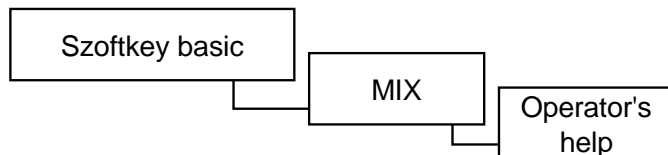
In the sidebar, select the **Win-s** Softkey, then select the icon in the pop up window. The **List of windows - NCTuser** window will pop up, where you can find a list of the previously opened windows. In the lower Softkey line, select the **Save Windows** tab.

Save changes: To keep the changes you just made, press **OK** on the **Prog.settings - NCTuser** window.

13.1.6. Setting the Help Text

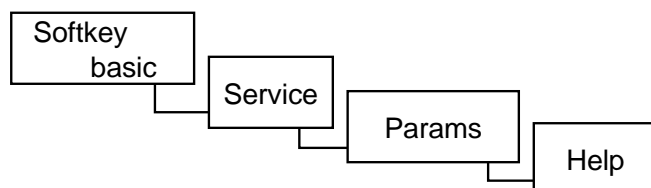
Besides the printed version of the operation manual, the help text can be also reached from the control panel through the following command chain:

SK > Mix > Operator's help



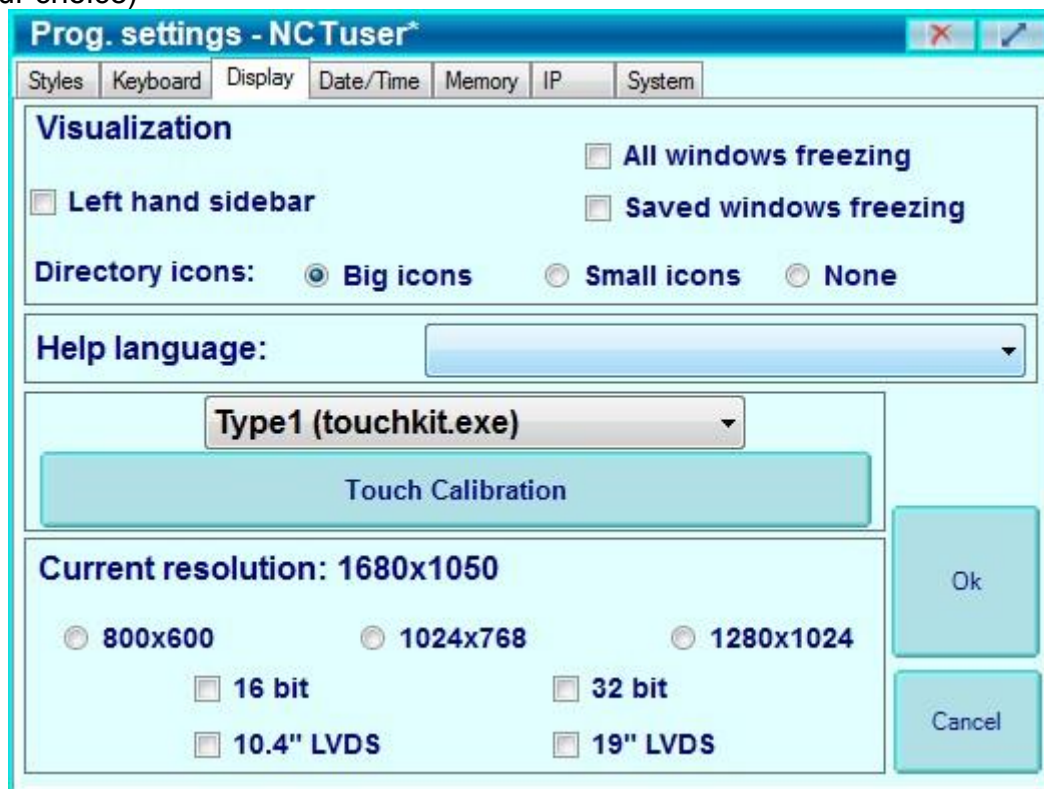
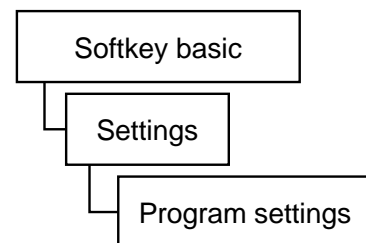
Additional help for the parameter description is possible also:

SK > Service > Params > Help



The languages of the help text can be set on the display tab. The set language for the help is independent from the basic user language of the CNC control. You can set the help text through the following command chain:

SK > Setting > Prog.setting > Display tab (use the Help language window and scroll down to the language of your choice)



Save changes: To keep the changes, press **OK** in the lower right corner of the window.

13.1.7. Calibration of the Touch screen

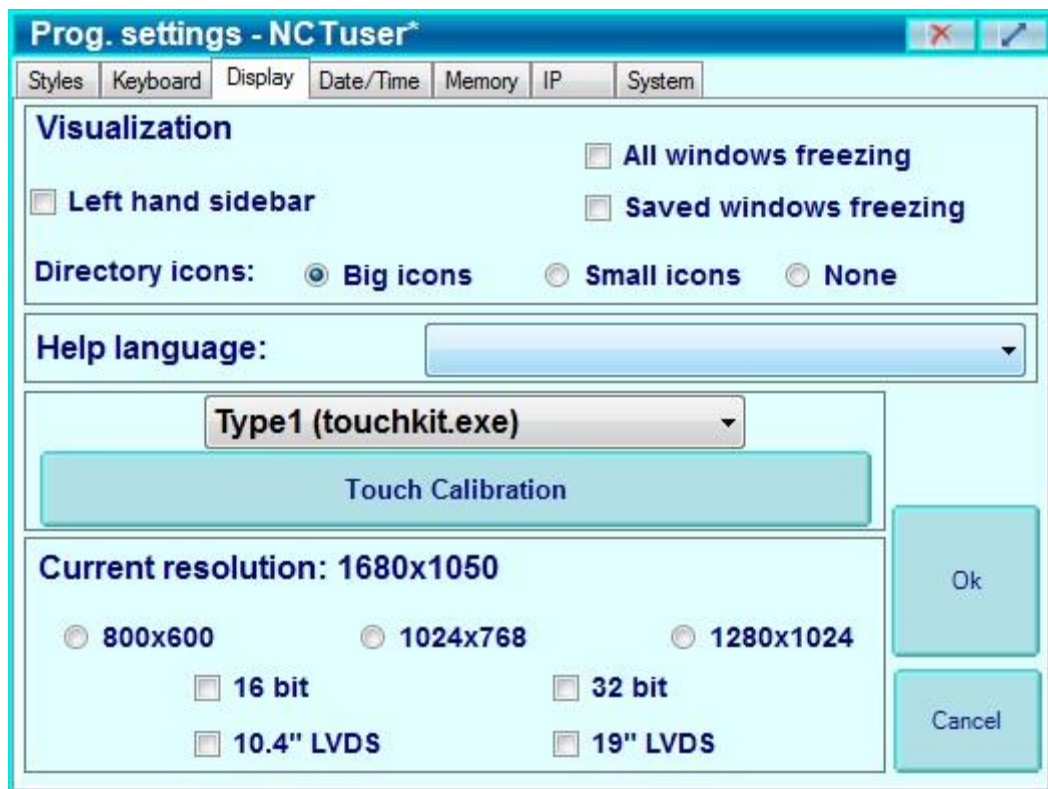
1. SK > Setting > Prog.setting

Softkey basic

Settings

Program settings

2. In the window, select the **Display** tab



3. Press the blue **Touch Calibration** button.

4. A new menu will then pop up: **TUB1: channel**.

5. You can now choose calibration on either 4 points or 25 points.

6. Once a calibration point number has been chosen, touch the flashing red X appearing on the screen until you hear a sound. The flashing will then stop and the X will turn green. Once every cross has turned green, the menu will appear again. This used to determine the accuracy of the calibration.

7. Select **Exit** to quit the calibration.

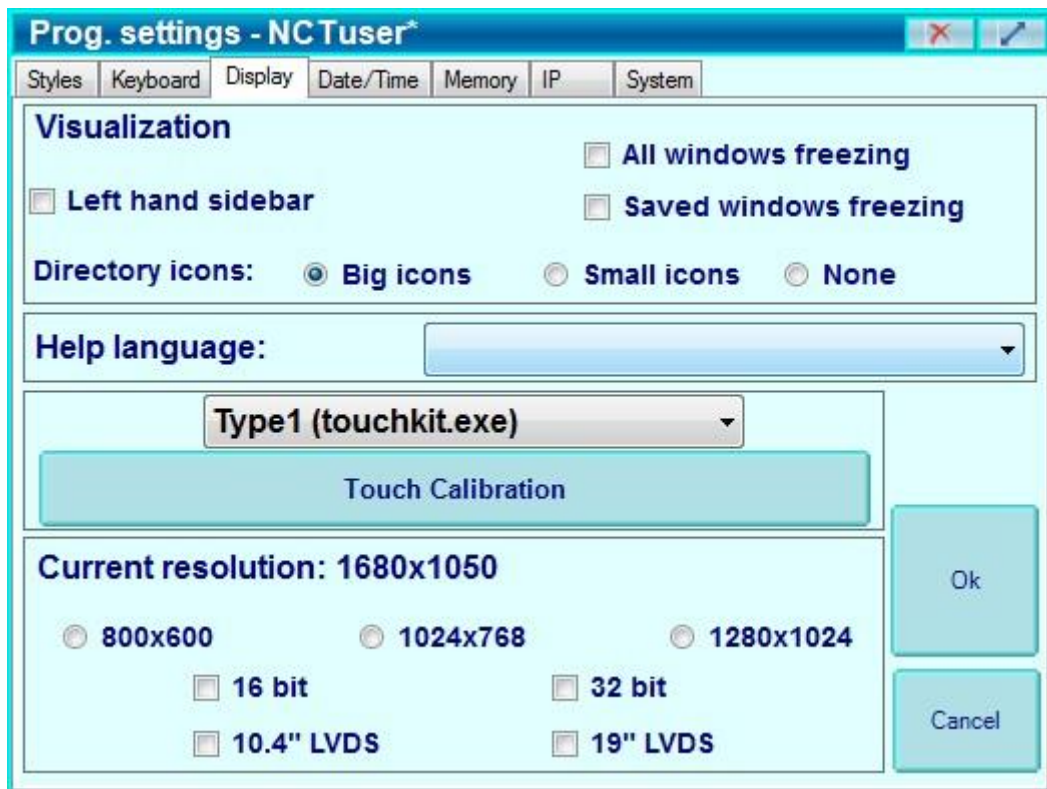
12.2.8. Setting the Resolution of the Screen

SK > Setting > Prog.setting > Display

Softkey basic

Settings

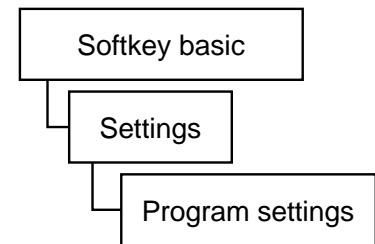
Program settings



In the bottom field of the window, you can find the possible screen resolution options. The already selected resolution will be displayed next to **Current resolution**. If you need to resize the screen or pick a different resolution, simply select any of the other boxes that contain the desired resolution. You can also select the size of the screen. You will need a full-authorization access for modifying the resolution.

13.1.8. Setting the time and date

SK > Setting > Prog.setting > Display > Date/Time



You will find two different setting fields separated by a frame. On the left side, you can set the date and time. In the **Current time** field, the actual current time is displayed, shown by the hours, minutes, and seconds. To set it, you can use either the arrows **UP** and **DOWN** or you can directly overwrite them by touching (and selecting) the numbers. In the time field, the hour, minute, and second can also be selected separately. You can set the selected part by using the arrows on the side of the field.

14. Channels

One of the most important strengths of the **NCT 2xx** is the support of the multi-channel control unit. The maximum number of the channel is 8, but it may be different depending on the software version that is being used. The several channels mean several logical CNC controls in one physical unit. Every channel is a separate CNC control, executing its own technological program, and it fully capable of being adjusted independent of the all other channels.

For each channel, you will find a separate tool correction and zero point tables, a macro variables table, a time and part counter, a position display, a single block window, program list and an FST window. Here are the most common switches that modify the program execution in each separate channel: single block execution, optional block skip, optional stop, program test, machine lock function, dry run, block return, block repeat, and MST functions locked.

In case of a multichannel machine, the following addendums are needed for the operation:

14.1. Handling of the Windows

The content of certain windows depend on the channel, so a certain window can belong to every channel. For example, each channel has its own position display. In the header of such windows, you can see the number of the channel, which is related to the content of the window. It is possible to display multiple windows at the same time, in different channels.

14.2. Position display of channels

On an **NCT 2xx** control unit that is connected to a multichannel machine, you will be able to collect positions, coordinates and other data, channel by channel in a separate window.

- 1) Windows can be called by the **Position** button in **SOFTKEY** line. The position windows of those channels will appear and the channel number will be found in the 4th field of the first line of the status display.

								Ch01: CH1 Turn	12:50:20
AUTM 1				mm					

- 2) Press **New window** in the **SOFTKEY** line. A new window will appear, possibly hiding the old one, but still containing the exact same content.



- 3) Press the **CH ++** button in the bottom of the sidebar.
- 4) The content of a new open window will change to the content of the next channel.
- 5) Grab this window by its header and the new window can be put next to the old one, so that you can see the position of the two channels at the same time.
- 6) You can select different windows to display the content of the different channels by selecting the **CH++** button. We can show many more position display windows by adding **New window**.

- 7) Other windows are dependant on different channels and can be displayed at the same time.

14.3. The **START** button

On an **NCT 2xx** control unit that is connected to a multichannel machine, there is only one **START** button. Once pressed, all the channels in automatic operation mode will start but only those that have a valid part program.

14.4. **STOP** button

When you press the **STOP** button, all the feed movements will stop and all the channels in automatic operation mode will go to a **STOP** state.

14.5. Multi-channel Program Execution in Automatic Operation Mode

After having created several part programs, you have to select them for automatic execution by picking a program for each channel. It is very similar to the basic operation with one channel control.

1. To select the first program to run, use the following command chain: **SK > Program > Directory1 (or Directory2)**. A directory will then appear.
2. Select the program you wish to run, then select **Run /AUTO** on the **SOFTKEY** line and the channel of the machine will appear.
3. Select the channel on which you wish to run the given program.
4. The program will be loaded to the automatic execution storage on the given channel. At the end of the status bar of the channel that you have chosen, the name and the access way of the program will appear.
5. In the **SOFTKEY** line, the channel's number won't disappear and the same program can be designated for other channels. ***
6. Set the other programs to the selected channels.



7. Press the **Auto Operation Mode** button in any of the channels that you wish to run.



8. Upon pressing the **Cycles Start** button, the program execution will start on all channels in automatic operation mode which contain valid part programs.

14.6. Operation Mode Change

You can define different operation modes for each channel. In the basic state, when you press an operation button, all the channels will get the same operation mode. It is mainly defined by the machine tool builder. If you press a channel selection button and an operation mode selection button at the same time, it is common for only the selected channel to take the operation mode. For the exact use of your machine, please contact the machine tool builder.