Operator's Guide 03/2004 Edition

sinumerik

SINUMERIK 840D/810D
HMI Embedded

SIEMENS

SIEMENS

SINUMERIK 840D/810D

HMI Embedded

Operator's Guide

Valid for

Control	*Software version*
SINUMERIK 840D powerline	7
SINUMERIK 840DE powerline (export version)	7
SINUMERIK 840Di	3
SINUMERIK 840DiE (export version)	3
SINUMERIK 810D powerline	7
SINUMERIK 810DE powerline (export version)	7

03.04 Edition

SINUMERIK® Documentation

Printing history

Brief details of this edition and previous editions are listed below.

The status of each edition is shown by the code in the "Remarks" columns.

Status code in the "Remarks" column:

A .... New documentation.
B .... Unrevised reprint with new Order No.
C .... Revised edition with new status.

Edition	Order No.	Remarks
02.01	6FC5298-6AC00-0BP0	A
11.01	6FC5298-6AC00-0BP1	C
11.02	6FC5298-6AC00-0BP2	C
03.04	6FC5298-6AC00-0BP3	C

Additional information is available on the Internet under:
http://www.siemens.com/motioncontrol

This publication was produced with with WinWord V 7.0 and Designer V 4.0.

Other functions not described in this documentation might be executable in the control. This does not, however, represent an obligation to supply such functions with a new control or when servicing.

We have checked that the contents of this documentation correspond to the hardware and software described. Nonetheless, differences might exist and therefore we cannot guarantee that they are completely identical. The information contained in this document is, however, reviewed regularly and any necessary changes will be included in the next edition. We welcome suggestions for improvement.

Order No. 6FC5298-6AC00-0BP3
Printed in Germany

Siemens Aktiengesellschaft

Preface

Structure of the documentation

The SINUMERIK documentation is organized in 3 parts:

- General Documentation
- User Documentation
- Manufacturer/Service Documentation

Target group

This document is designed for machine tool users. The document describes in detail all the technical facts an operator needs to understand how to operate the SINUMERIK 840D, 840Di, 810D control systems.

Hotline

If you have any questions, please get in touch with our hotline:

A&D Technical Support

Phone.: +49 (0) 180 5050 - 222

Fax: +49 (0) 180 5050 - 223

If you have any questions about the documentation (suggestions, corrections etc.), please send a fax or e-mail to:

Fax: +49 (0) 9131 98 - 2176

E-mail: motioncontrol.docu@siemens.com

Fax form: See the reply form at the end of the document.

Internet address

http://www.siemens.com/motioncontrol

SINUMERIK 840D powerline

Since 09.2001 the

- SINUMERIK 840D powerline and
- SINUMERIK 840DE powerline (export version)

have been available with improved performance. The hardware description below contains a list of the available **powerline** modules:

References: /PHD/ SINUMERIK 840D Configuration Manual

SINUMERIK 810D powerline

Since 12.2001 the

- SINUMERIK 810D powerline and
- SINUMERIK 810DE powerline (export version)

have been available with improved performance. The hardware description below contains a list of the available **powerline** modules:

References: /PHC/ SINUMERIK 810D Configuration Manual

Export version

Function	810DE	840DE
Five axis machining package	–	–
Handling transformation package, (five axes)	–	–
Multi-axis interpolation, (> four axes)	–	–
Helical interpolation, 2D+6	–	–
Synchronized actions, stage 2	–	O[1)]
Measurements, stage 2	–	O[1)]
Adaptive control	O[1)]	O[1)]
Continuous dressing	O[1)]	O[1)]
Utilization of compile cycles, (OEM)	–	–
Sag compensation, multi-dimensional	–	O[1)]

- Function not available
1) restricted functionality

Standard scope

This Operator's Guide describes the functionality of the standard scope. Extensions or changes made by the machine tool manufacturer are documented by the machine tool manufacturer.

Please contact your local Siemens office for more detailed information about other SINUMERIK 840D, 810D publications, and publications that apply to all SINUMERIK controls (e.g. universal interface, measuring cycles, etc.).

Other functions not described in this documentation might be executable in the control. This does not, however, represent an obligation to supply such functions with a new control or when servicing.

Validity

Catalog NC 60 is definitive in terms of the validity of the functions /BU/ Machine Tools, catalog NC 60

Structure of the descriptions

Wherever sensible and possible, all functions and operating options have been described using the same internal structure. By structuring the explanations in different information levels, we have made it possible for users to specifically access the required information.

Explanation of symbols

Function

This theoretical part serves as a learning document, particularly for those who are new to the world of NC applications. Important information about understanding operating functions is offered to users at this point.
You should work through the manual at least once to get an overview of the operational scope and performance capability of your SINUMERIK control.

Operating sequence

This point shows the sequence of key operations which need to be executed. Any inputs or additional information required in the individual operating steps are described next to the pictures of the keys.

Notes

This symbol is always displayed in this document to draw your attention to an important item of information in a more complex description.

Machine manufacturer

For safety reasons some functions are locked against unauthorized access. The machine manufacturer can influence or change the described functional behavior. Always refer to the information and specifications provided by the machine manufacturer.

Further information

For safety reasons some functions are locked against unauthorized access. The machine manufacturer can influence or change the described functional behavior. Always refer to the information and specifications provided by the machine manufacturer.

Reference notes

You will find this symbol wherever further information on a particular topic can be looked up in specialist literature.
A complete list of available literature is included in the Appendix of this Operator's Guide.

Ordering data option

In this documentation you will find the symbol shown on the left with a reference to an ordering data option. The function described will only be executable if the control contains the designated option.

Warning Information

The following warning notices with varying degrees of significance are used in the document:

Danger
Indicates an imminently hazardous situation which, if not avoided, **will** result in death or serious injury or in substantial property damage.

Warning
Indicates a potentially hazardous situation which, if not avoided, **could** result in death or serious injury or in substantial property damage.

Caution
Used with the safety alert symbol indicates a potentially hazardous situation which, if not avoided, **may** result in minor or moderate injury or in property damage.

Caution
Used without safety alert symbol indicates a potentially hazardous situation which, if not avoided, **may** result in property damage.

Notice
Used without the safety alert symbol indicates a potential situation which, if not avoided, **may** result in an undesirable result or state.

Principle

Your SIEMENS 840D, 810D is designed using state-of-the-art technology in conformity with recognized safety regulations, standards and specifications.

Supplementary devices

The applications of SIEMENS controls can be expanded by adding special additional devices, equipment and expansions supplied by SIEMENS.

Personnel

Only **properly trained, authorized, reliable personnel** must be allowed to use this equipment. No-one without the necessary training must be allowed to operate the control, even temporarily.

The **areas of responsibility** assigned to personnel involved in setting up, operating and maintaining the equipment must be clearly **specified** and their compliance **verified**.

Procedure

Before the control is started up, personnel responsible for its operation must have read and understood the Operator's Guides. The company using this equipment is also obliged to carry out **continuous monitoring** of the overall technical condition of the equipment (with a view to identifying externally visible defects and damage as well as changes in the operating behavior of the control).

Servicing

Repairs to equipment may only be carried out by **personnel specially trained and qualified** for the application in question in accordance with the provisions specified in the maintenance and servicing guides. All relevant safety regulations must be followed.

The following are deemed as **improper usage** and **exclude the manufacturer from all liability**:

Any usage or application incompatible with or beyond the scope of the items specified above.

Cases where the control is operated **in a technically imperfect condition**, without due provision for safety considerations and/or hazards, or in contravention with any of the instructions in the relevant documentation.

Cases where faults which could affect safety are not remedied **before** starting up the control.

Any **modification, bypassing** or **decommissioning** of equipment on the control whose intended purpose is to ensure proper functioning, unrestricted use of equipment and/or active and passive safety.

Unforeseen danger can arise with reference to:

- life and limb of personnel,
- the control, machine or other assets of the owner and the user.

■

Contents

Introduction

1.1 The product SINUMERIK 840D/810D

General

SINUMERIK 840D and 810D are CNC control systems (**C**omputerized **N**umerical **C**ontrol) for machine tools.

You can implement the following basic functions (for a machine tool) via the operator panel front of the CNC control:

- Creating and adapting parts programs.
- Execution of parts programs,
- Manual control,
- Input and output of parts programs and data,
- Editing of data for programs,
- Display and precisely recover alarms,
- Editing machine data,
- Creating communication connections between 1 or more PCUs, or 1 or more NCs
  (m:n, m-HMI units and n-NCK/PLC units).

The user can call up all the functions via the user interface.
The user interface consists of:

- Display devices such as a screen, light diodes, etc.
- Operator controls such as keys, switches, hand wheels, etc.

This Operator's Guide describes the procedure for PCU/PC with the **HMI Embedded Software, SW 6.5**

The HMI Embedded software is installed in the PCU as standard when supplied from the factory.

Read Chapter 2 "Operator Components" carefully before proceeding with further chapters.
All subsequent chapters are written on the assumption that you have done so!

1.2 Switching control system on and off

Function

Switching the control ON

A variety of methods can be employed to switch on the power supply to the control system or to the whole station, therefore:

Machine manufacturer

Please refer to the information and specifications provided by the machine manufacturer!

After the control has been switched on, the "Reference point approach" display or another basic display programmed by the machine manufacturer will appear.

Switching the control OFF

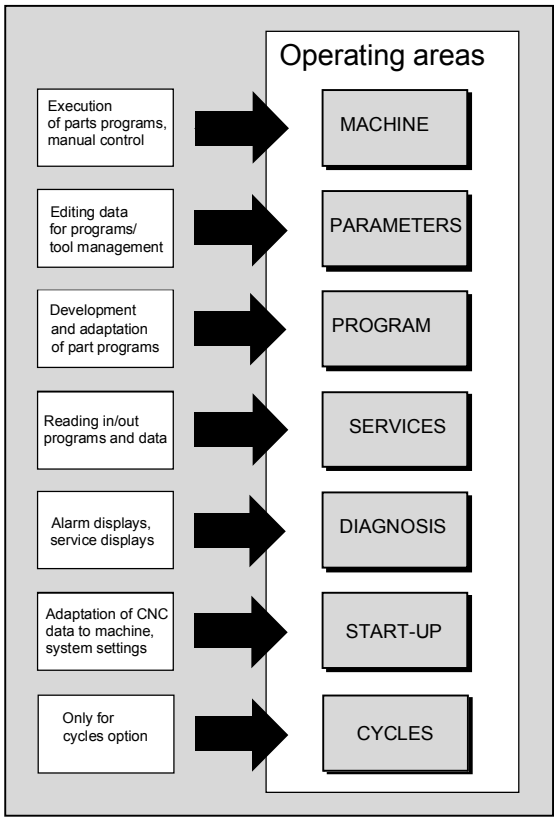
Please follow the instructions for switching off the control or the entire system!

Machine manufacturer

Please refer to the information and specifications provided by the machine manufacturer!

1.3 Operating areas

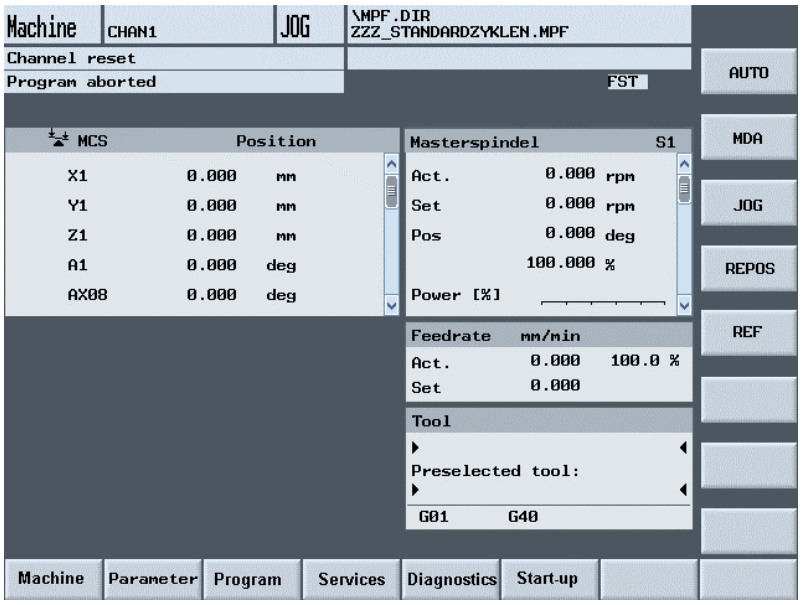
The basic functions are grouped in the following operating areas in the control (in gray boxes):



Operating sequence

MENU SELECT

When you press the "Area switchover" key, operating areas are displayed on the horizontal softkey bar and operating modes are displayed on the vertical softkey bar. You can use this key to go to the area menu bar from any location in the menu hierarchy if you wish to select another operating mode or a different operating area.



MENU SELECT

By pressing the "Area switchover" key twice, you can toggle between the operating areas last selected, e.g. between the "Parameters" and "Machine" areas.

■

Operator Components/Operating Sequences

2.1 Operator panels

The recommended hardware base for HMI Embedded is a SINUMERIK 840D/810D with PCU 20 or PCU 50.

You can use one of the following operator panels for the PCU:

OP 010
OP 010C
OP 010S with OP 032S full CNC keyboard
OP 012
OP 015 with 19" full CNC keyboard

TP 012 and TP 015A (only with customer-specific software)

OP 010 Operator Panel

A Display
B Alphanumeric keypad
Correction/cursor keys

1 Machine area key
2 Recall (Return)
3 Softkey bar (horizontal)
4 ETC key
(menu extension)
5 Area switchover key
6 Softkey bar (vertical)

All keys are described in the following sections.

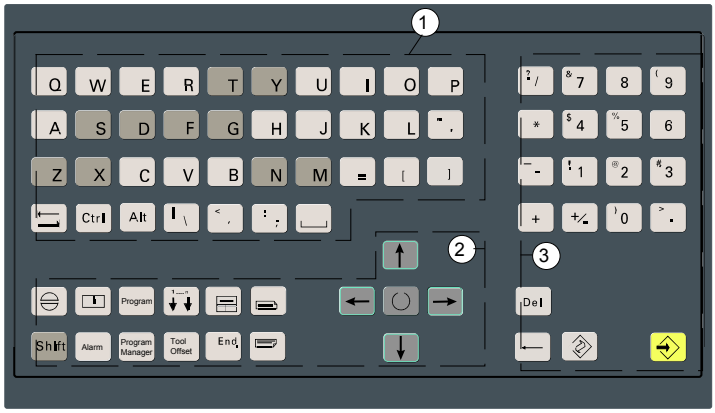
OP 010S Operator Panel

A Display

1 Machine area key
2 Recall (Return)
3 Softkey bar (horizontal)
4 ETC key
(menu extension)
5 Area switchover key
6 Softkey bar (vertical)

OP 032S with full CNC keyboard

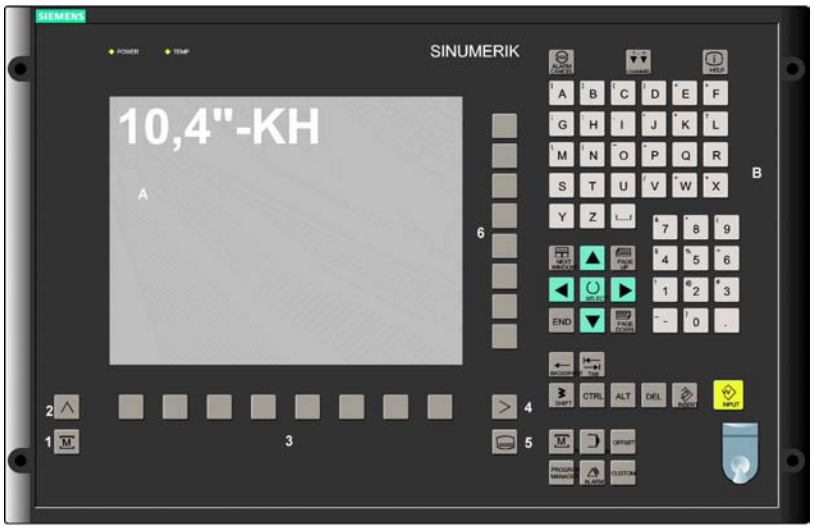
1 Alpha pad
2 Correction/cursor pad
with control keys
3 Numeric pad



OP 010C Operator Panel

A Display
B Alphanumeric keypad
Correction/cursor keys

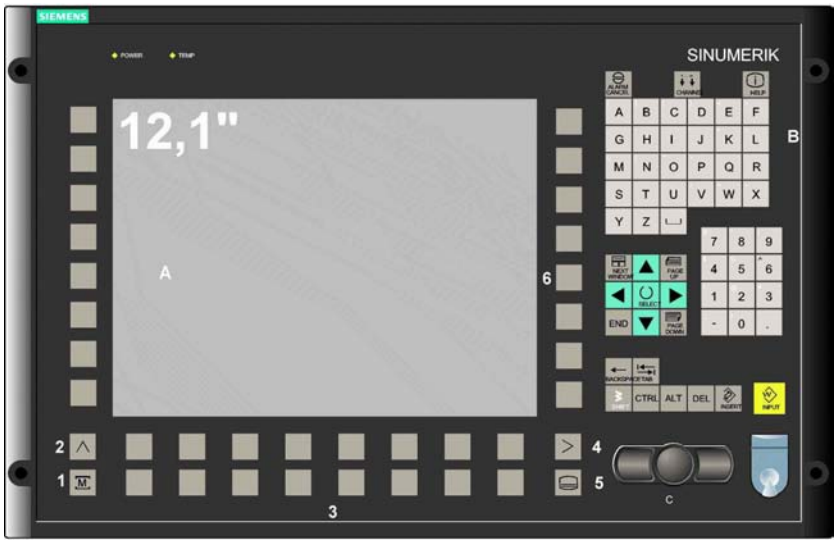
1 Machine area key
2 Recall (Return)
3 Softkey bar (horizontal)
4 ETC key
(menu extension)
5 Area switchover key
6 Softkey bar (vertical)



OP 012 Operator Panel

A Display
B Alphanumeric keypad
Correction/cursor keys
C Mouse and mouse keys

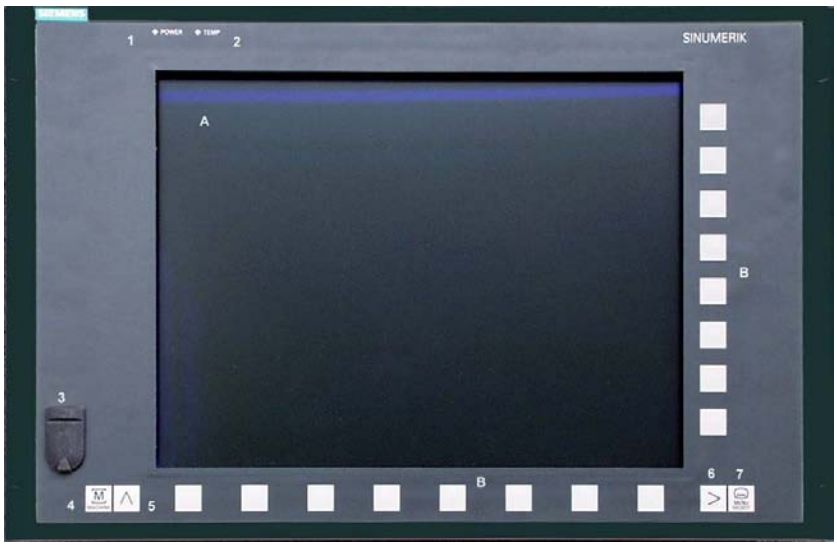
1 Machine area key
2 Recall (Return)
3 Softkey bar (horizontal)
4 ETC key
(menu extension)
5 Area switchover key
6 Softkey bar (vertical)



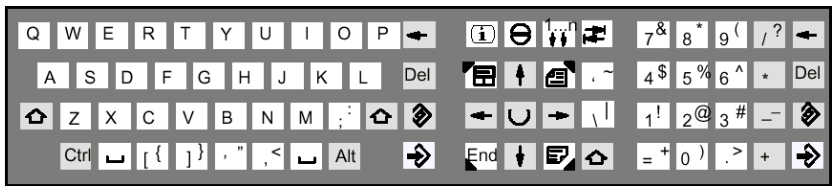
OP 015 Operator Panel

A Display
B Softkeys
1 Status LED: Power
2 Status LED: TEMP*)
3 Front USB interface
4 Machine area
5 Recall
6 ETC key
7 Area switchover key

*) Wear will probably be high when the TEMP LED is lit.

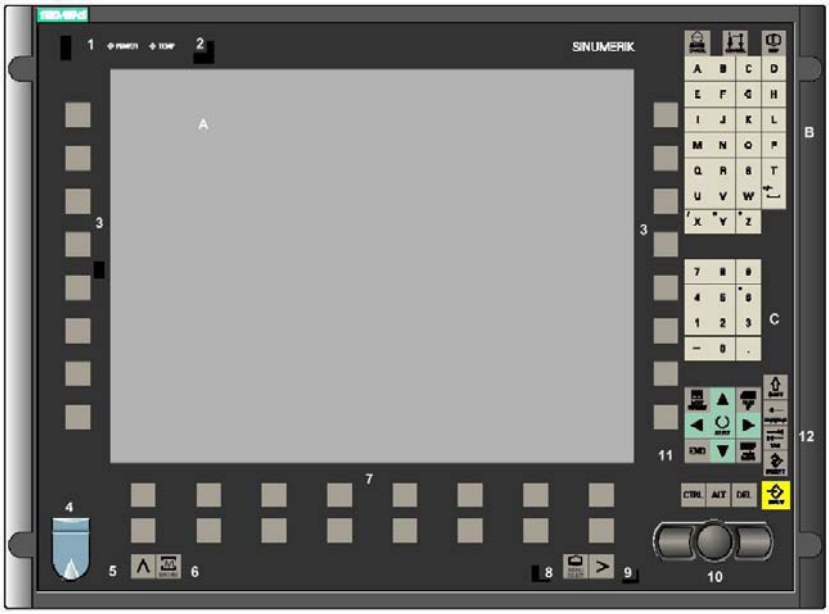


With full CNC keyboard



OP 015A Operator Panel

A Display
B Alpha pad
C Numeric pad
1 Status LED: Power
2 Status LED: TEMP*)
3 Softkeys and direct keys
4 Front USB interface
5 Recall
6 Machine area
7 Softkeys
8 Area switchover key
9 ETC key
10 Mouse
11 Cursor pad
12 Control keypad



*) Wear will probably be high when the TEMP LED is lit.

Notice

If a high contrast picture is displayed on the TFT display for more than an hour, there is a danger of irreversible display damage due to the so-called "burn-in effect".
In order to avoid this undesired effect, the screen saver must always be active.

References

/IAM/IM2, Installation Guide HMI Embedded, Chapter 5 Switching on Display Darkening Switch

2.1.1 Operator panel keys

The elements of the operator panel keyboard and the symbols used to represent them in this manual are shown and explained below.
The keys marked with an * correspond to the key symbols in US layout.

MACHINE
Direct branch to the "Machine" operating area.

Recall key
Return to the next higher menu. Recall closes a window.

ETC key
Expansion of the softkey bar in the same menu.

MENU SELECT
You can call the basic menu from any operating area by pressing this key. Pressing the key twice in succession changes from the current operating area to the previous one and back again.
The standard basic menu branches into the following operating areas:
1. Machine
2. Parameters
3. Program
4. Services
5. Diagnostics
6. Start-up

ALARM CANCEL
By pressing this key, you can acknowledge the alarm marked by this cancel symbol.

CHANNEL
In a configuration with several channels, it is possible to switch between channels (switch from channel 1 to channel n).
When a "Channel menu" is configured, all existing communication links to other NCUs plus the associated channels are displayed on softkeys.
(See also Section "Channel switchover")

HELP
This key displays explanatory information about the current operating status (e.g. support for programming, diagnosis, PLC, alarms).
The letter "i" displayed in the dialog line indicates that information is available.

SHIFT
Depress the Shift key to enter the uppermost character shown on the dual input keys.

CTRL

Use the following key combinations to navigate in the process plan and in the G code editor:

Ctrl + Pos1: Jump to the start.

Ctrl + End: Jump to the end.

Ctrl + C: **Copy** a selected area to the same global memory. If nothing is selected, the content of the entire field is saved in the buffer.

Ctrl + X: Copy a selected area to the same global memory and **cut**.

CTRL +V: **Insert** the contents from the global memory to the cursor position.

ALT

Blank, empty character

BACKSPACE

- Delete the value in the input field from the right.
- Delete the character after the cursor in insertion mode.

DEL

- Delete the value in the parameter field
- Delete the character marked by the cursor in insertion mode.

INSERT

- Switch to tables and input fields in the editing mode (in this case the input field is located in the insert mode) or
- UNDO function on the table element and input fields (when leaving a field with the edit key, the value is not accepted, but rather reset to the previous value = UNDO).
- Activate pocket calculator mode

TAB

Tab key

INPUT

- Accepts an edited value
- Open/close a directory or program

NEXT WINDOW

If several windows are displayed on the screen, it is possible to make the next window the active one using the window selection key (the active window has a thicker border).
Keyboard input, e.g. the page keys, is possible only in the active window.

Cursor up

Move between different fields or lines.

PAGE DOWN

You "page" down by one display.
In a part program you can scroll the display down (towards end of program).
With the page keys you scroll the visible/displayed area of the window that is active. The scroll bar indicates which part of the program/document/... is selected.

Cursor to the left

- Move between different fields or lines.
- Switch to the higher level directory.

SELECT

This key has the same function as the "Alternat." softkey.

- Selection key for values set in fields and selection lists labeled with this key symbol.
- Activate or deactivate a field:

☒ = active	◉ = active
☐ = inactive	◯ = inactive
Multiple selector button (you can select several options or none)	Single selection button/option (only one option can be active at a time)

- Switch Select mode on and off in the editor

Cursor to the right

- Move between different fields or lines.
- Open a directory or program.

END

- The cursor in the editor moves to the end of the line in the opened page with this key.
- Rapid positioning of the cursor on a group of related input fields.

Cursor down
Move between different fields or lines.

PAGE UP
You "page" up by one display. With the page keys you scroll the visible/displayed area of the active window. The scroll bar indicates which part of the program/document/... is selected.

PROGRAM (only OP 010 and OP 010C)
Open the "Program" operating area.
This key has the same function as the "Prog. edit" softkey.

OFFSET = (only OP 010 and OP 010C)
Tool management
Open the "Tools/zero offset" operating area.
This key has the same function as the "Tool zero point" softkey

PROGRAM MANAGER (only OP 010 and OP 010C)
Open the "Program" operating area.
This key has the same function as the "Program" softkey.

ALARM (only OP 010 and OP 010C)
In the main diagnostic display, open the "Messages/Alarms" overview of alarms.
This key has the same function as the "Alarm list" softkey.

CUSTOM
The customer configures this key.

Initiates a PCU reset

2.2 Machine control panels

Standard turning machines/milling machines

Actions on the machine tool, for example traversing the axes or program start, can only be initiated via a machine control panel.

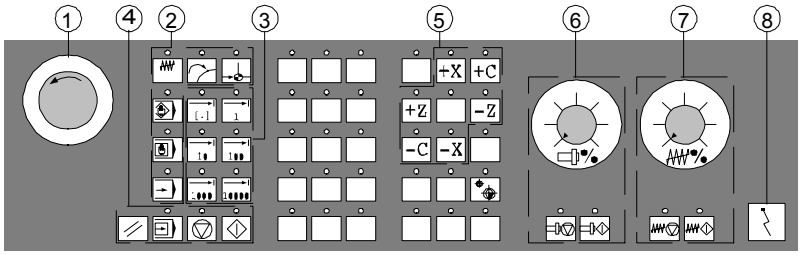
The machine tool can either be equipped with a standard machine control panel from SIEMENS or with a specific machine control panel from the machine-tool manufacturer.

The machine control panel described in this document is the panel supplied by SIEMENS. If you are using another machine control panel, please consult the operating instructions of the machine-tool manufacturer.

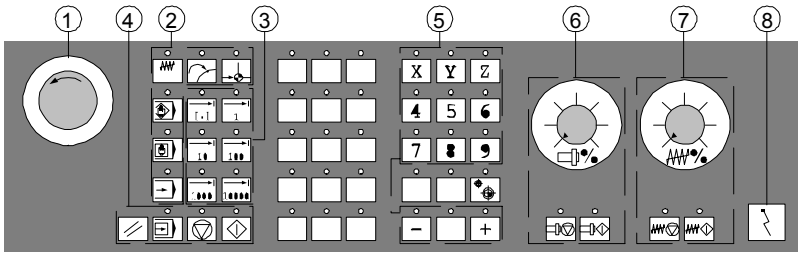
The standard machine control panel from SIEMENS is equipped with the following operator controls:

1 EMERGENCY STOP key
2 Operating modes (with machine functions)
3 Incremental traverse/Increment
4 Program control
5 Direction key with rapid traverse override

6 Spindle control
7 Feedrate control
8 Key switch

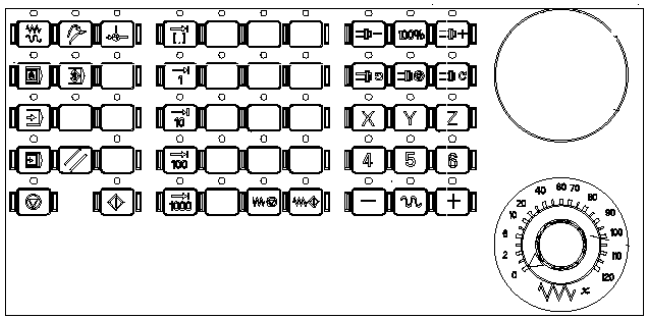


Machine control panel for turning machines



Machine control panel for milling machines

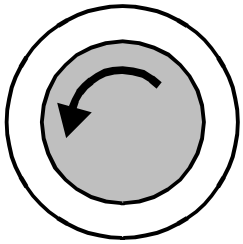
Machine control panel OP 032S



Machine control panel OP 032S

2.3 Machine control panel keys

2.3.1 Emergency stop key



Press this red key in emergency situations:
1. If life is at risk,
2. When there is the danger of machine or workpiece being damaged.

An EMERGENCY STOP generally shuts down all drives with the greatest possible braking torque in a controlled manner.

Machine manufacturer

For details of other or additional reactions to an EMERGENCY STOP: Please refer to the information and specifications provided by the machine manufacturer.

2.3.2 Operating modes and machine functions

If you press a "Mode key", the corresponding mode is selected if permissible, and all other modes and functions are deselected.

The active mode is signaled and confirmed by the associated LED which lights up.

Jog
Select Machine Manual operating mode. Conventional traversing of the axes via:
- continuous motion of the axes using the direction keys or
- incremental motion of the axes using the direction keys or
- the handwheel.

Teach In
Creation of programs in interactive mode with the machine in "MDA" mode.

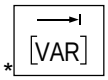
MDA (Manual Data Automatic)
Select "Machine Auto" operating mode.
Control of machine through execution of a block or a sequence of blocks. The blocks are entered on the operator panel front.

Automatic
Select "Machine Auto" operating mode.
Control of machine through automatic execution of programs.

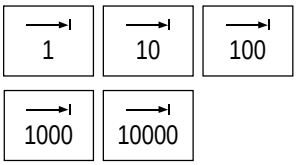
Inc keys

You can activate the Inc functions in conjunction with the following modes:

- Operating mode: "Jog"
- Operating mode: "MDA/Teach In"



VAR (Incremental feed variable)
Incremental traverse with variable increment size (see "Parameters" operating area, setting data).



Inc (Incremental Feed)
Incremental traverse with preset increment size of 1, 10, 100, 1000, 10000 increments.

The way that the incremental value is evaluated depends on the setting in the machine data.

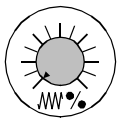
See machine manufacturer's specifications.

Machine functions

REPOS
Repositioning
Reposition, re-approach contour in "Jog" mode.

Ref Point
Approaching a reference point
Approach the reference point (Ref) in "Jog" mode.

2.3.3 Feedrate control



Feedrate rapid traverse override (feedrate override switch)

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

Settings
0%, 1%, 2%, 4%, 6%, 8%, 10%, 20%, 30%, 40%, 50%, 60%, 70%, 75%, 80%, 85%, 90%, 95%, 100%, 105%, 110%, 115%, 120%

Feed Stop

- Execution of the active program is stopped,
- The axis drives are shut down,
- The associated LED illuminates as soon as feed stop has been accepted by the control,
- FST (= feed stop) appears in the header (program control display)

Example:
– In the "MDA" operating mode an error is discovered during execution of a block.
– Tool change is to be carried out.

Feed Start

If you press the "Feed start" key:

- the part program in the current block is continued,
- the feed is accelerated to the value defined in the program,
- the associated LED illuminates as soon as feed start has been accepted by the control.

Axis keys (for turning machines):

Traverse the selected axis (X.... Z).

Axis keys (for milling machines):

Select axis (X…. 9) for traversing

in a positive direction with the "+" key or

in a negative direction with the "-" key

Rapid
Move axis at rapid traverse (fastest speed).

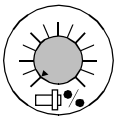
Machine manufacturer

- The specified increments and control range apply to standard machines.
- Increments and control range can be modified by the machine tool manufacturer to suit specific applications.
- Feedrate/rapid traverse and the values for the feedrate override switch settings (if the feedrate override switch is also operative in rapid traverse) are defined in a machine data.

Please refer to the information and specifications provided by the machine manufacturer.

WCS/MCS
Switch between the workpiece coordinate system (WCS = work) and machine coordinate system (MCS = machine).

2.3.4 Spindle control



Spindle override (spindle speed override switch)

- The rotary switch with latch positions allows you to increase or decrease the programmed spindle speed "S" (equivalent to 100%).
- The set spindle speed value "S" is output as an absolute value and a percentage in the "Spindles" display (vertical softkey in basic display).

Control range:
50% to 120% of programmed spindle speed

Increment:
5% between latch positions

Spindle stop

When you press the "Spindle Stop" key:
- the spindle is decelerated down to zero speed and
- the associated LED illuminates upon "Spindle Stop".

Example:
- to change a tool.
- to enter S, T, H, M functions during setup.

Spindle start

When you press the "Spindle Start" key:
- the spindle speed is accelerated to the value defined in the program and
- the associated LED lights up as soon as "Spindle Start" has been accepted by the control.

Machine manufacturer

– The specified increment and the control range apply to standard machine data (MD). These MD can be changed by the machine-tool manufacturer to suit the application.
– The maximum spindle speed and the values for the spindle speed override position are defined in the machine data and setting data (see information supplied by the machine-tool manufacturer).

2.3.5 Key switch

SIEMENS key switch

The key switch on the SINUMERIK 840D, 810D has 4 settings to which protection levels 4 to 7 are assigned.

Machine manufacturer

Functions can be assigned to key switch positions by the machine manufacturer. Using machine data it is also possible to set access to programs, data, and functions to suit the user's requirements.
The key switch has three different colored keys which can be removed in the specified positions:

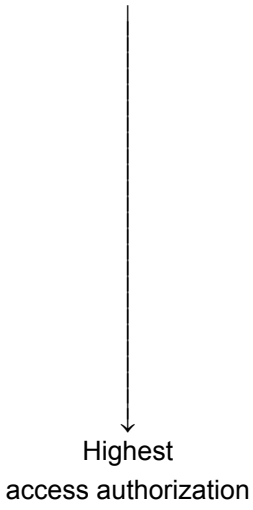
Key positions

Position 0
No key
Protection level 7

Position 1
Key 1 **black**
Protection level 6

Position 2
Key 1 **green**
Protection level 5

Position 3
Key 1 **red**
Protection level 4



Changing access rights

The screen is not automatically updated after a change in access authorization (e.g. when the key switch position is changed), but only when the screen is next refreshed (e.g. on closing and opening a directory).
The currently valid access authorization is checked every time a function is executed.

If the PLC is in the stop state, the input image of the machine control panel is not scanned. For this reason the key switch positions are not evaluated during start-up.

Passwords

Access authorization can also be set through the input of three passwords in the basic display of the "Start-Up" operating area.
If the password is set, the key switch positions are irrelevant.

References

/IAD/Installation & Start-Up Guide 840D or
/IAC/Installation & Start-Up Guide 810D

2.3.6 Program control

Cycle Start
Start execution of a selected program:
The associated LED illuminates.

Cycle Stop
Stop processing of a running parts program:
The associated LED illuminates.
Press the "Cycle Start" key to continue processing.

Single Block
Execute a part program block by block You can activate the "Single Block" function in "Automatic" and "MDA" modes. If single block is activated, the associated LED on the machine control panel lights up. If single block execution is active,

- a stop in the cycle is displayed on the screen (in the program control display line),
- (in interruption status) in the channel operational messages line - the text "Stop: Block ended in SBL mode" is output,
- the current block of the part program is only processed once you press the "Cycle Start" key,
- the processing is stopped after execution of one block,
- you can execute the next block by pressing the "Cycle Start" key again.

You can deselect the function by pressing the "Single Block" key again.

The current status is shown in the program control display (global machine status display, see Chapter 2 Screen Layout)

This function is dependent on the settings under "Program control" in the Machine operating area.

Reset

- Execution of the current program is aborted.
- Messages from the cycle monitoring are deleted (except for POWER ON, Cycle Start and "Acknowledge alarm" alarms).
- The channel is shifted to "reset" state, i.e.,
  - The NC control remains synchronized with the machine.
  - The control is in its initial state and ready for a new program run.
    (See also)

References /FB1/K1: Description of Functions Mode Group, Channel, Program Operation

2.3.7 Standard PC keyboard MF-II

A standard PC keyboard can be connected. However, a machine control panel is required in addition to this keyboard.

The special function keys on the operator panel front can also be used with the standard PC keyboard. The following table shows the keys on which the horizontal and vertical softkeys are mapped:

Standard PC Keyboard	F1	F2	F3	F4	F5	F6	F7	F8	F9	F10	F11	F12
with SHIFT	vertic Soft. 1	vertic Soft. 2	vertic Soft. 3	vertic Soft. 4	vertic Soft. 5	vertic Soft. 6	vertic Soft. 7	vertic Soft. 8		MACHINE		CUSTOM
without SHIFT	horiz Soft. 1	horiz Soft. 2	horiz Soft. 3	horiz Soft. 4	horiz Soft. 5	horiz Soft. 6	horiz Soft. 7	horiz Soft. 8		MENU SELECT	CHANNEL	HELP

Standard PC keyboard	Esc	Insert	Home	Page Up	Page Down	Enter	Tab		Num Block 5	1	3	7	9
with SHIFT													
without SHIFT	ALARM CANCEL	INSERT	NEXT WINDOW	PAGE UP	PAGE DOWN	INPUT	END						
switched off Num Block									SELECT	PROGRAM	OFFSET	PROGRAM MANAGER	ALARM

Caution

The standard PC keyboard does not meet the requirements (EMC) of a SINUMERIK control. For this reason, it should only be used for installation and servicing purposes.

2.3.8 Mouse operability with HMI Embedded Win32

The HMI Embedded Win32 software with the PCU 50/70/ PC hardware can also be operated with a mouse.

Right mouse button functionality
Right clicking the mouse has the same effect as activating the area switchover key.

Left mouse button functionality

Single-click	• The dialog cursor is positioned on a closed IO field; input starts with the first character. • If the IO field is open, it closes and the input is not validated. When you enter a value, the old value is deleted. • If you click a character in an open field, the mouse pointer is positioned after the last character in the string.
Double-click	The field is opened in input mode, the editor cursor is positioned on the first character.
Drag depressed mouse button	The characters are highlighted according to the mouse position.
Click the scroll bar	Paging though
Double-click a program	Open the program
	Click characters in open part program
Click selection field	Make next selection
Click check field	Invert field, make next selection

Note This function is not supported on the PCU 20.

2.4 Screen layout

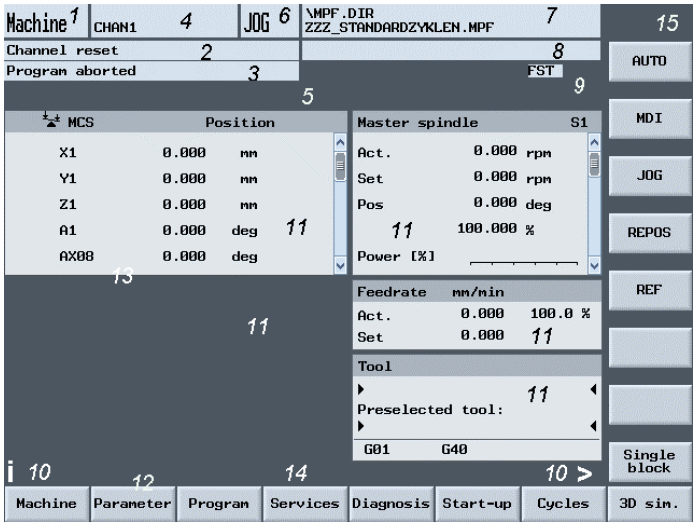
2.4.1 Overview

Softkeys
Keys to which functions are assigned by means of a menu bar displayed on the screen.

- It is possible to access further menu levels via the horizontal softkeys in any operating area. Each horizontal menu item has a vertical menu bar/softkey assignment.
- The vertical softkeys are assigned functions for the currently selected horizontal softkey.

A function is called up by pressing one of the vertical softkeys. The assignments of the vertical softkey bar can change if further subsidiary functions are classified under a function.

Global machine status display



1. Operating areas
2. Channel status
3. Program status
4. Name of channel
5. Alarm and message line
6. Operating mode
7. Program name/path of the selected program
8. Channel operational messages
9. Program control
10. Additional explanatory information (Help) recallable
    - i Information can be displayed via the i-key
    - ^ Recall: Return to higher-level menu
    - > ETC: Expansion of the softkey bar in the same menu.

11. Working windows, NC displays
    The working windows (program editor) and NC displays (feedrate, tool) available in the selected operating area are displayed here.

    From SW 6.2
    Positional data in the working windows show the diameter symbol ∅ before the unit if the axis is currently the traverse axis and if the tool coordinate system is set. If diameter programming is disabled with DIAMOF, the symbol preceding the unit is no longer visible.

12. Dialog line with operator notes. Operator notes for the selected function appear here (if available)
13. Active window
    The selected window is displayed by its own frame. The window header display is inverted. Data entered on the operator panel front apply to this window.
14. Horizontal softkeys
15. Vertical softkeys
    The softkey functions available in the selected operating area are displayed in the horizontal and vertical softkey menus (corresponding to F1 to F8 on the full keyboard).

2.4.2 Global machine status display

1 Operating areas	The currently selected operating area is displayed (Machine, Parameters, Program, Services, Diagnosis, Start-Up).
2 Channel status	The current channel status is displayed, • Channel reset • Channel interrupted • Channel active
3 Program status	The current status of the running part program is output: • Program aborted • Program running • Program stopped

4 Channel name	Name of channel in which program is running.
5 Alarm and message line	• Alarms and messages or • References that were inserted in the part program with the MSG command (if no alarms are active)
6 Operating mode display	The currently selected operating mode, i.e. Jog, MDA or AUTO (automatic) is displayed.
7 Program name	This program can be executed with Cycle Start.
8 Channel operational messages	1 Stop: No NC ready 2 Stop: No mode group ready 3 Stop: Emergency Stop active 4 Stop: Alarm active with stop 5 Stop: M0/M1 active 6 Stop: Block ended in SBL mode 7 Stop: Cycle Stop active 8 Wait: Read-in enable missing 9 Wait: Feedrate enable missing 10 Wait: Remaining dwell time: ... secs 11 Wait: Aux. funct. ackn. missing 12 Wait: Axis enable missing 13 Wait: Exact stop not reached 14 Waiting for positioning axis 15 Waiting for spindle 16 Waiting for other channel 17 Wait: Feedrate override to 0% 18 Stop: Error in NC block 19 Waiting for NC blocks from external 20 Waiting due to SYNACT instruction 21 Wait: Block search active 22 Wait: No spindle enable 23 Wait: Axis feedrate value is 0 24 Waiting for tool change acknowledgement 25 Waiting for gear stage change 26 Waiting for position control 27 Waiting for thread cut 28 Wait: 29 Waiting for punching 30 Waiting for safe operation

31 Stop: Channel not ready
32 Stop: Oscillation active
33 Stop: Axis replacement active (block change inhibited because axis replacement in progress)
34 Waiting for axis container rotation
35 Wait: AXCT axis active as slave axis
36 Wait: AXCT axis active as master axis
37 Wait: AXCT axis changing to follow-up
38 Wait: AXCT internal status change in axis
39 Wait: AXCT axis drive disable
40 Wait: AXCT axis overlaid motion active
41 Wait: AXCT axis, axis replacement active
42 Wait: AXCT axis interpolator active
43 WAIT_FOR_CC_ENABLE: Waiting for compile cycle
44 Waiting for access to system variable
45 Stop; Serupro has found search target and the NCK has stopped.
ERUPRO is the abbreviation for SEarchRUn by PROgramtest; it is a new type of block search. SERUPRO is activated via PI service "_N_FINDBL" parameter == 5;
SW $[[SW410000]] and later.
46 = Stop; ESR activated
47 Wait: Axis container rotation waiting for spindle stop
48 Wait: Axis container rotation waiting for MD data match (New-Config)
49 Waiting for axis replacement: Axis currently coupled
50 Waiting for axis replacement: Liftfast active
51 Waiting for axis replacement: New-Config active
52 Waiting for axis replacement: Axis container rotation active
53 Waiting for axis replacement: Waitp active
54 Waiting for axis replacement: Axis is currently in another channel
55 Waiting for axis replacement: Axis is currently PLC axis
56 Waiting for axis replacement: Axis is currently reciprocating
57 axis
58 Waiting for axis replacement: Axis is currently jog axis
59 Waiting for axis replacement: Axis is currently command axis
60 Waiting for axis replacement: Axis is currently OEM axis
Waiting for axis replacement: Axis is currently slave axis in master-value coupling
61 Waiting for axis replacement: Axis is currently coupled-motion axis
62 Waiting for axis replacement: Axis is currently coupled slave axis

9 Program control — Functions that have been activated are displayed (settable via "Program control").

2.4.3 Program control display

Only functions that have been activated are visible (settable via "Program control"). The channel status line is displayed irrespective of the selected menu.

SKP Skip block

References /PG/ Programming Guide Fundamentals, Chapter 2

DRY
Dry run feedrate

Traverse movements are performed with the feedrate value set in the setting data "Dry run feedrate". The dry run feedrate function replaces the programmed travel commands.

ROV
Rapid traverse override

The override switch for the feedrate also applies to the rapid traverse feedrate.

M01
Programmed stop

When this function is active, processing of the program is stopped at every block in which miscellaneous function M01 is programmed.
The message "Stop: M00/M01 active" is then displayed on the screen. You restart processing with the Cycle Start key. if the function is not active, then the miscellaneous function M01 (from the part program) is **disregarded**.

M101
Halt at cycle end

According to option set.

DRF
DRF selection

When the "DRF" function is active, DRF offsets are applied.

PRT
Program test

In program test mode, set point outputs to the axes and spindles are disabled. The set point display "simulates" the traverse movements.

FST
Feed stop

The activated feed stop is displayed.

This function is not activated/deactivated under Program Control, but via the Feed Start/Feed Stop keys on the machine control panel.

2.5 General operating sequences

Keys A range of keys and menus is provided in the various operating areas. The function of the keys and menus is the same in all operating areas.

Functions The following section describes functions which you can select in several operating modes.

2.5.1 Program overview and program selection

Function

After selection of a workpiece or program overview, individual workpieces or programs can be enabled or disabled for execution.

Operating sequence

"AUTO" is selected in the "Services" or "Machine" operating area.
The appropriate channel is selected.
The channel is in reset state.
The workpiece/program to be selected is in the memory.

An overview of all workpiece directories/programs that exist is displayed.

Position the cursor on the desired workpiece/program.

Select the workpiece/program for execution:

Select program

The name of the selected workpiece is displayed on the screen in field "Program name" at the top. The program is then loaded.

2.5.2 Selecting and opening directory/file

The "Direction keys" can be used to position the cursor on the desired directory/file.
If you enter a character on the alphanumeric keyboard, the cursor moves to the first name that begins with the character.

Open/close directory:

You can open a directory with the "Input" key.

Press "Recall" to close it again.

Open file:

You can open a file with the "Input" key if you wish to edit it in the ASCII editor. The editor is opened automatically.

2.5.3 Editor

Function

The editor is used to create, update and modify programs and texts (e.g. alarm texts).

Machine manufacturer

Please note the information supplied by the machine tool manufacturer!

Opening a second editor window:
You have the option of opening a second program and displaying this alongside the one already open on the screen.

Operating sequence

Press the ETC key. The horizontal softkey menu displays the "2nd file" softkey. After pressing the softkey, select another program and open it using the input key. Now both editor windows are displayed alongside each other on the screen.

Use the "Next window" key to toggle between the windows.

Enlarge window

If you press the "Enlarge window" softkey, the program where the cursor is displayed over the whole screen.

Decrease window

If you press the "Decrease window" softkey, both editor windows are displayed next to each other.

Close editor

Close editor

The current program overview appears again when you activate the "Close editor" softkey.
Changes are saved in the program immediately when the cursor leaves the modified line.

Editor settings

You can define the following values in the "Editor Settings" window:

- Enable a newly created program automatically when it is closed,
- Skip the "Line feed",
  See also Chapter 6: Selective Program Protection: RO
- Display hidden lines (HD) in the program,
  See also Chapter 6: Hidden program lines: display HD
- Shorten positioning time within a large file (e.g. in mold making).
  You can select a special procedure as well as specify file size up to 58 KB.

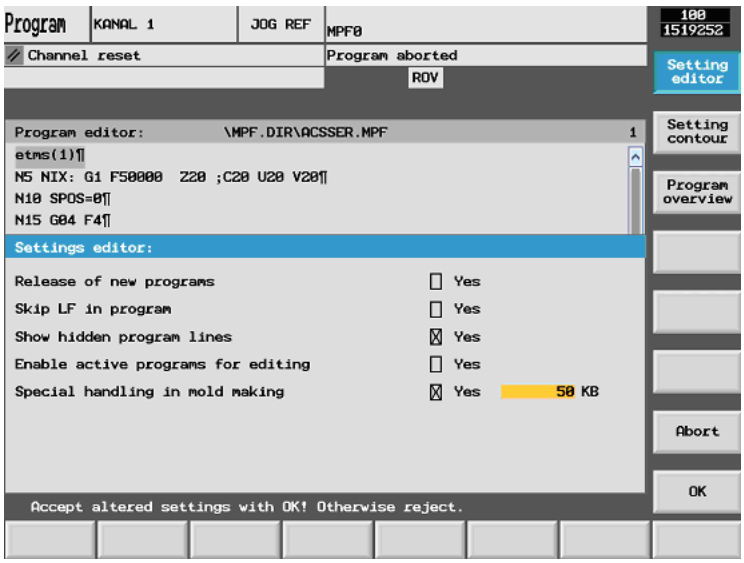
Operating sequence

Settings

If you press the horizontal softkey "Settings"

Editor settings

and the vertical softkey "Editor settings" you get the following window:



Contour settings

Contour programming

The following settings are possible for contour programming:

- Specify the contour end, the entered line (e.g. "End of contour") is displayed after each contour.
- Specify the technology, turning or other
- Display the softkeys for contour elements as symbols or text.

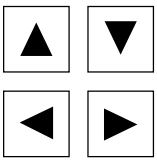
For information on what to do next, see Chapter 6: Free contour programming

2.5.4 Changing menu window

Using the "Window selection" key, you can toggle between individual menu windows. You only need to do this if you wish to enter data on the operator panel front. The focus changes to the selected menu window (the header and border of the active window appear in a different display format).

Scroll in menu window:

If the contents of a window cover several screen pages, you can use the "Page" keys to scroll up or down through the information.
A scroll bar indicates that the contents of the window extend beyond the visible display.



Position cursor in menu window:

You can position the cursor at the desired point in the menu window with the "Direction keys".

2.5.5 Editing inputs/values

If you wish to edit inputs/values, the corresponding key is always displayed automatically on the right of the input field. The following input fields are available:

1. Option buttons (single selection button/multiple selection button):
You can use the "Select" key to activate or deactivate a selection field.

Multiple selector button (you can select several options or none)	Single selection button/option button (only only one option can be active at a time)
☒ = active	◉ = active
☐ = not active	○ = not active

2. Input fields:
Position the cursor on the input field and start to type. When you begin typing, you automatically switch to Insert mode.

Always confirm your input with the "Input" key. The value is accepted.

To explicitly alter an existing value, press the "Insert" key to switch to input mode.

Enter the value or word (e.g. file name, etc.) on the alphanumeric keypad.

In some fields you can choose between several preset values using the "Select" key.

- The editor displays only the characters which can be input via the operator panel front keyboard.

2.5.6 Confirming/canceling input

Confirming input:

OK

Your inputs are accepted when you press softkey "OK". The selected function is executed. The window is closed, and you return to the call menu level.

Canceling input:

Abort

Your inputs are rejected if you select the "Abort" softkey. The selected function is aborted. The window is closed and you return to the menu level from which the window was called.
The response is the same on return from a function (vertical softkey bar).

The "Insert" key can also have an "Undo" function if you abandon the input/modification you have just entered. The cursor remains positioned in the currently selected field.

Switch from the horizontal menu level back to the call menu level.

2.5.7 Editing a part program in the ASCII Editor

Function

The ASCII editor provides you with the following functions:

- Switch between insert and overwrite mode
- Mark, copy, delete block
- Paste block
- Position cursor / find/replace text
- Create contour (programming support)
- Configure cycle parameters (drilling, milling, turning)
- Start simulation
- Recompile (cycles, free contour programming)
- Renumber blocks
- Change settings

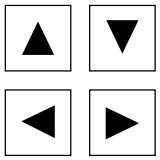
Further information

A part program selected in the NC can generally only be edited when the channel is in the reset state.
When a part program is selected and the relevant channel in the "Channel Reset" state, the program can be fully edited.

Operating sequence

The following functions are fully enabled in the Program operating area but only partially in operating areas Machine and Services.
In the Machine operating area, the ASCII editor is called via the program editor, in Services, by selecting a file in the file manager.

You have selected the file you wish to edit in the directory and pressed the "Input" key,
the vertical softkey bar changes.
Your selected file is opened in the text editor.



Cursor block:
Use the "Direction keys" to position the cursor in the text.

You can page up and down using the "Page" keys.

The character on which the cursor is positioned is deleted.

Press the "Delete" key to delete the character to the left of the cursor.

Press the "Input" key to end a block. The characters "L_F" ("Line Feed") are automatically generated. The block will not be accepted for processing without a line feed character.

Vertical softkeys

Overwrite

Overwrite

The softkey toggles between insert and overwrite modes.

Mark block

Mark block

When you press this softkey, the vertical softkey bar changes. The area marked by the cursor is selected.

Copy block

The selected block is copied to a buffer. It remains stored in the buffer even if another part program is selected.

Delete block

The selected block is deleted.

Mark block or SELECT

You can cancel select mode with softkey "Mark block" or "Select". A block generated with the support function is not automatically selected.

A limit is applied to the block area which can be selected. When the limit is reached the following warning appears in the dialog line: "Buffer limit for selection reached".

Paste block

Paste block

The softkey pastes the cut or copied block from the buffer into the text in front of the cursor position.

Find/Go to...

Find/ Go to ..

The window "Find/Go to..." is opened.
You can select functions for positioning and searching via the vertical softkeys:

The following search methods are available:

Start of Program or CTRL NEXT WINDOW

- to the beginning of the part program (cursor on the first character in the program),

End of Program or CTRL END

- the end of the part program (cursor on the last character in the program) and

Go to ..

- to position on a particular NC block with "Go to.."

Find

- or search for a particular character string with "Find".

"Go to...":
Enter the block number you are looking for.

- If the line being searched for contains an "**N**" or "**:**" you are taken to that block,
- If there is no block with the specified number, a message is output.

OK

Press softkey "OK" or the "Input" key to position the cursor on the block number/line number of your choice.
The "Go to..." window is closed.

Abort

If you abort positioning, the process is stopped and the window closed.

Find

"Find"
Enter the character string you wish to find.

The string you enter is sought downwards from the current cursor position, the find result appears as highlighted text.

Find next or INPUT

You can start a new search by pressing softkey "Find next" or the "Input" key.

Replace or INPUT

Enter the new text with the "Replace" softkey.
The text found is replaced by the "replacement text". The new text is replaced when you press "Input". Every time you press "Input", a new find and replace process is started.

Abort

If you abort Find + Replace, the process is stopped and the window closed. You are in "Edit mode" again.

Save file

Save file
Changes are saved in the file loaded in the editor.

Further information

Please note that the changes to programs stored in the NC memory take immediate effect.

Horizontal softkeys

Free contour programming

Support | New contour

You can call the free control programming function via softkeys "Support" and "New contour".

Accept contour

The part program block is inserted in the part program with the appropriate parameters.

Cycle parameterization

The following programming support functions are available:

- Drilling, milling, turning (cycles)
- Contouring (free contour programming)

Drilling | Milling | Turning

You can call up the relevant cycle compilation via the vertical softkeys Drilling, Milling, Turning.

OK

The part program block is inserted in the part program with the appropriate parameters.

Example:

```
CYCLE81 (110, 100, 2, 35)
```

References /PGZ/Programming Guide Cycles

Simulation

Simulation

The simulation function is called.
Pressing the Cycle Start key starts the graphics simulation.
See Chapter 6: Section on "Program Simulation"

Cycle recompilation

If program steps (cycle/contour) have already been parameterized but must be changed, you can display and edit the parameter values with their meanings.

▼

Position the cursor in the text editor on the line with the program step (contour/cycle) whose parameters you wish to change.

Recom-pilation

A screen form containing the parameter settings for the selected cycle/contour appears on the screen.

Change the parameters.

OK

The part program block is automatically inserted in the part program with the new parameters.

Renumber

The block numbering in the program shown in the editor is executed again according to the values defined under the "Settings" softkey. Once the softkey is selected another window opens. Specify the "Block number" and the "Increment".

Machine manufacturer

The coordinate system and the technology to be used are set via the machine data. Please refer to the information and specifications provided by the machine manufacturer.

References

/IAM/IM2, Installation & Start-Up Guide HMI Embedded

2.5.8 Channel switchover

It is possible to switch between channels when several are in use. Since individual channels may be assigned to different mode groups, a channel switchover command is also an implicit mode switchover command.
If the selected channel is located on another NCU (m:n link), the HMI Embedded is switched implicitly to this NCU.
When a "Channel menu" is configured, all existing communication links to other NCUs plus the associated channels are displayed on softkeys.

Channel states

The following three channel states can occur in each mode:

1. **Channel reset**
   The machine is in the initial state, e.g. after power-on or after end of program. The initial state is defined by the machine-tool manufacturer in the PLC program.

2. **Channel active**
   A program has been started, program execution or reference point approach is in progress.

3. **Channel interrupted**
   The running program or reference point approach has been interrupted.

In this context a program can be a main program, subprogram, cycle or a series of NC blocks.

There are 3 different switchover levels:
1. Switch to next channel.
2. Switch over configured channel group/channels (1 NC).
3. Switch to another NC (with m:n link involving several NCs).

2.5.9 m:n communication links

General

An m:n link means that m-HMI Embedded-units and n-NCU/PLC-units are linked together. This does not however mean that all possible links are active.

An HMI Embedded is linked with only one NC at a given point in time (cf. 1:1 link) and communicates only with this NC. With an m:n link, the link can also be switched over to another NC. Using the channel switchover key and channel menu, you can switch the MMC over to another link.

You can change to the channel menu in all operating areas by activating the channel switchover key. The only change is to the horizontal and vertical softkeys.
Use the horizontal softkeys to select a channel group (max. 8),
8 links to channels on different NCUs can be set up in each channel group.

All current communication links and associated symbol names are listed in display "Channel menu".

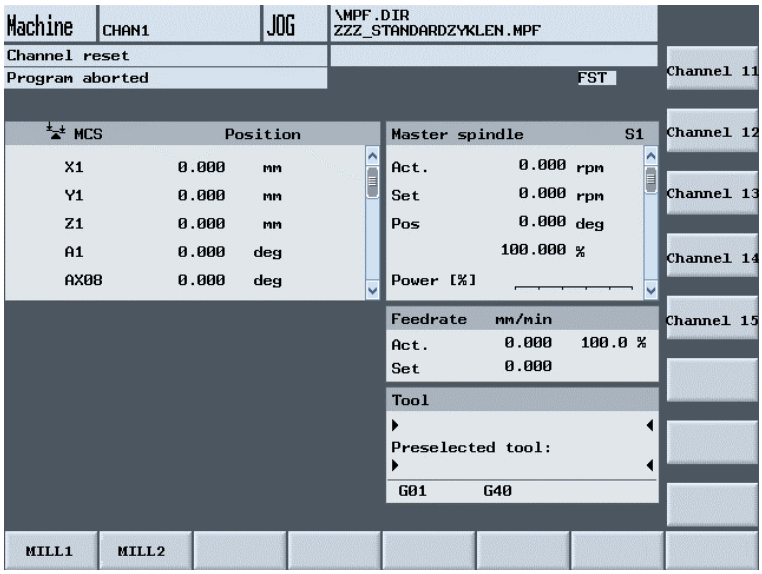
Important
Only two links may be active simultaneously on one NCU.

References

/FB2/B3: Several Operator Panels and NCUs

Function

You can set up a link in any operating area between the HMI unit and the connected NCU/PLC units via the operator interface.



Activate the channel switchover key. The currently existing connection is displayed by means of the highlighted softkeys (horizontal, vertical) if the channel menu is active.

Channel switchover

You can switch to other channels by means of the vertically arranged softkeys.

Group switchover

Use the horizontal softkeys to switch over to a different group; the channels of the current group are now displayed on the vertical softkeys. Switchover to another channel (and if necessary to another NC) only takes place upon activation of a vertical softkey.

NC switchover

You can change to another NC via the vertical softkeys if the channel is not on the current NC.

Further information

- When you select a vertical softkey, you are selecting a channel and potentially an NC as well.
- Channels that are configured in the channel menu, but defined as a channel gap in the corresponding NC, will not be displayed.

2.5.10 Pocket calculator

Precondition:
The cursor is positioned on an input field or input/output field.

You can switch to **Pocket calculator mode** with the equals key.
If you enter a basic arithmetic symbol (+, –, /, *) followed by a value (e.g. 13.5) in this mode and

and then press the input key, the next value to be entered is calculated with the previous value.
If the input/output field is opened with the input or equals key, the editor is in insert mode; if the field is opened directly with a character, the editor is in write-over mode.

Help display
If pocket calculator mode is active, you can call a help display which explains how to operate the input editor by pressing the Info key.

2.5.11 Input of fits

Function
The letter "F" ("f") is used to identify a fit. The letter is followed by the diameter/length specification. This is followed by the tolerance class and the tolerance quality identifier.
The arithmetic mean is calculated from the upper and lower limiting value and displayed in the field.

Note

Angle specifications are not supported.

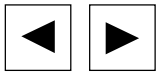
Example:

F20H7 This input stands for 20H7
or: F20h7 This input stands for 20H7
or:
F = fit
20 = Diameter/length specification (1... 500 mm)
H (h) = tolerance class (tolerance classes for drill holes and shafts: A, B, C, D, E, F, G, H, J, JS, T, U, V, X, Y, Z, ZA, ZB, ZC)
7 = Tolerance quality identifier (01... 18, DIN standard 7150 limits the area!)

Restrictions:

1. The diameter/length must be specified as an integer.
2. Quality identifiers 6, 7, 8 are available for class J.
3. For class j, only qualities 5, 6, 7 and 8 are available for diameter 1...3 mm.
4. Only qualities 8...18 are available for classes P, R, S; T, U, V, X, Y, Z, ZA, ZB, ZC, p, r, s, t, u, v, x, y, z, za, zb, zc.
5. Qualities CD, EF, FG, cd, ef, fg are only available for up to 10 mm.
6. According to the standard,
   classes T, t are only available for diameters > 24 mm,
   classes T, t are only available for diameters > 14 mm,
   classes Y, y are only available for diameters > 18 mm,

Recompilation of the value is not possible.



Upper/lowercase notation:

The operator panel does not have separate keys for lowercase letters, but you can use the selection key to change between upper and lowercase notation.

Position the cursor on a letter in the input field and press the selection key to toggle between upper and lowercase notation.
When you have finished, press the ENTER key to confirm your input.

2.5.12 Blue screen error status screen

In the event of a system crash, an error status screen displays the current system information. The screen shows a brief description of the error and displays any relevant process registers.
You will find various options for saving system information in the following documentation:

References: /IAM/ Installation & Start-Up Guide HMI Embedded (IM2)

Operating Example

3.1 Typical operating sequence

To provide support for entry-level users or an orientation guide for others, this section uses a typical operating sequence (from control system power-up to back-up of a user-generated part program) to explain how the functions described can be located.

	Step	Described in Section
Setup	• Switch on machine	1.3
	• Homing	4.3
	• Clamp workpiece/blank	5
	• Select tools	5
	• Define workpiece zero for coordinate inputs	5.8
	• Enter tool offsets	5.2
	• Calculate speeds and feedrates	4.2
	• Define a reference point (scratching)	4.4
Enter/test a program	• Create a part program or read one in via an external Import data interface	6.4 7.5
	• Select a part program	4.6
	• Test a program (without a tool) - Start a part program (e.g. in single block) - Edit part program using program editing function or diagnostics guide/help	 4.2 2.6/ 4.6 8.1
	• Optimize a part program	4.2
Machining the workpiece	• Insert tool and run machining program	5.2
Storing a program	• Save a part program - on external storage devices - Read out via RS-232 interface	6.7 4.6 7.5

■

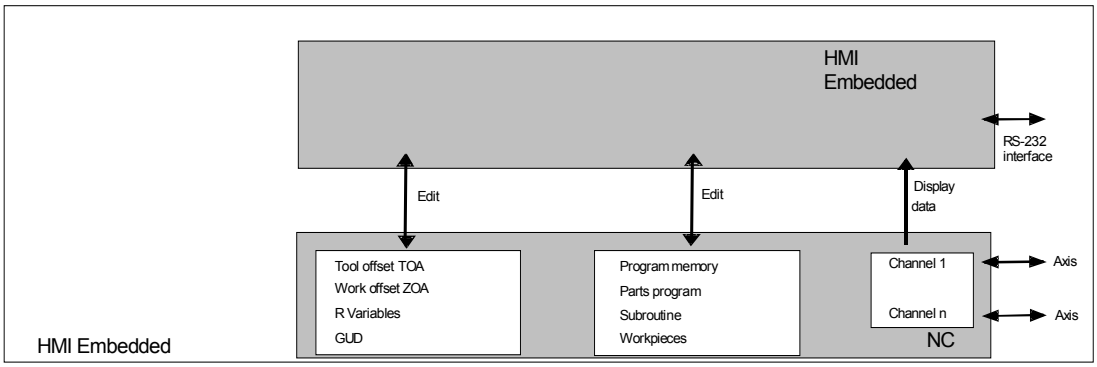
Machine Operating Area

4.1 Data structure of the NC control

Function

- NC with part program memory
- Part programs are processed in different channels (one program per channel).

HMI Embedded The data are always stored in the NC and changed directly in the NC.



4.1.1 Operating modes and machine functions

Function

The Machine operating area includes all functions and parameters which initiate actions on the machine tool or measure its status.
There are three different modes in this area:

- Jog: "Jog" is the mode required for manual operation and setting up of the machine. The setup functions provided are "Reference point approach", "Repositioning", "Handwheel" or "Traverse in preset increments" and "Redefine control zero" (preset).

- MDA: Semi-automatic operation
  Part programs can be set up and processed block by block in this mode. The tested blocks can then be stored in the part program.
- In “Teach In” mode, positions can be traversed and stored to generate motion sequences which are then stored in the MDA program.
- Automatic: Fully automatic operation
  Part programs are executed fully automatically in “Automatic” mode, i.e. programs are selected, started, corrected, selectively controlled (e.g. single block) and executed in Automatic.

Select machine area

You can switch to the "Machine" area at any time from any of the other operating areas simply by pressing the "Machine area" key.

When you switch on the control, it is usually in the operating area "Machine" in operating mode "Jog". (Please consult the machine manufacturer's documentation!)

Machine manufacturer

The state after power-up can be configured and might therefore deviate from the default.

Machine functions

In operating mode "Jog" you can select the following machine functions via the machine control panel or softkeys in the basic menu:

Inc (traverse in preset increments)

Repos (reposition at a defined position)

Ref (reference point approach to coordinate machine with the control zero points)

In "MDA" mode, it is possible to select "Teach In" (storage of motion sequences in a part program through position approach) by pressing the MCP key.

Preparation for production

To start actual production some preparatory measures must be taken:

1. Set up the tools and workpiece,
2. Traverse the tools or workpiece to the start position required in the setup plan.
3. Load the part program in the control memory,
4. Check or input the work/zero offsets,
5. Check or input the tool offsets.

4.1.2 Operating mode group and channels

Function

Every channel behaves like an independent NC in which a maximum of one part program can be processed.

- Control with one channel:
  Only one operating mode group exists.
- Control with several channels:
  Channels can be grouped to form several "operating mode groups."
  Example:
  Control with 4 channels, where machining is carried out in 2 channels and 2 other channels are used to control the transport of the new workpieces.

Mode Group1	Channel 1 (machining)
	Channel 2 (Transport)
Mode Group2	Channel 3 (Processing)
	Channel 4 (Transport)

Technologically related channels can be combined in an operating mode group.
Axes and spindles of the same mode group can be controlled by one or more channels.
An operating mode group is either in "Automatic," "Jog" or "MDA" operating mode, i.e. several channels of one operating mode group can not assume different operating modes at the same time.

4.1.3 Selection of operating mode / changing operating mode

Function

The defined modes for operating a SINUMERIK control system are Jog, MDA and Automatic. They are selected via the MCP or by means of softkeys.

Machine manufacturer

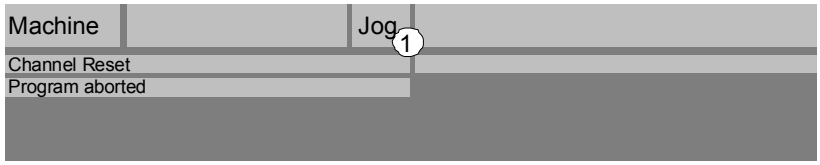
Whether the requested mode can be accessed and the manner in which it is accessed can be configured machine-specifically in the PLC program.

Operating mode change

Not all mode changes are permitted.
An error message is output if a mode change request is rejected by the system. The error message will indicate the error cause and possibly the remedy.

Operating sequence

Selection of mode
The selected mode is displayed in the mode field on the screen.



1 = operating mode

To select the modes

- Jog
- MDA
- Automatic

press one of the keys shown on the left on the machine control panel
or

the corresponding vertical softkey that you can access via the "Area switchover" key:

AUTO

MDA

JOG

- Automatic
- MDA
- JOG

When a mode is selected, the LED next to the selection key on the MCP lights up. The same status is signaled in the mode field on the screen.

Further information

The basic display of the selected mode appears on the screen.

If a mode change is not possible, please contact your installation engineer, the machine-tool manufacturer or our service personnel. In many cases, a mode change is enabled only for trained personnel for reasons of safety. To provide this type of protection, the control system offers a facility for disabling or enabling mode changes.

References /FB//K1: Mode Group, Channel, Program Operation

4.2 General functions and displays

4.2.1 Start/stop/abort/continue part program

Function

The following section describes how to start and stop part programs and to resume them after they have been aborted.

Operating sequence

"AUTO" mode is selected in the "Machine" operating area.

Precondition:
No alarms are pending.
The program is selected.
Feedrate enable is active.
Spindle enable is set.

Start part program:
The part program is started and executed.

Stop/abort part program:

The program run is interrupted, but can be restarted with “Cycle Start”

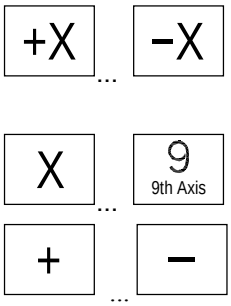
The current program is aborted.

Resume part program:
After a program interruption ("Cycle Stop") you can retract the tool in manual mode ("Jog") from the contour. The control saves the coordinates of the point of interruption. The distances traversed by the axes are displayed.

Repositioning:
Select the "Machine" operating area.

Select "Jog" mode.

Press the "Repos" key to reposition the tool on the contour.



Turning machine:
Press the "+X" or "–X" key.

Milling machine:
Select the axis to be traversed and

Press the "+" or "–" key.

Traverse the axes to the point of interruption.

4.2.2 Display program level

Function

If subprograms are called while a part program is running, the block numbers for the main program and subprograms together with their pass number (P) can be displayed.

Operating sequence

"AUTO" mode is selected in the "Machine" operating area.

When you press the "Program level" softkey, the "Program level" window appears instead of the "Current block" window. The softkey labeling changes to "Current block".

During execution of a part program, the block numbers for the main program and subprograms, together with their pass number (P), are displayed in the "Program level" window. The main level is always visible, a nesting depth of up to 12 subroutines can be displayed.

Current block

When you press the "Current block" softkey, the "Current block" window containing the program blocks of the current part program is displayed again.

4.2.3 Switching between the machine/workpiece coordinate system (MCS/WCS)

Function

The display can be toggled between the machine and workpiece coordinate systems by means of special key "MCS/WCS" on the MCP or via softkeys (depending on MCP model and user program). The actual position display for the distance-to-go and the corresponding axes change.

Machine axes

Machine axes are axes that actually exist on the machine and have been parameterized during installation.

Geometry axes and special axes

These are the axes programmed in the part program. Geometry axes and special axes are offset by the selected zero offset relative to the machine axes.
Three is the maximum number of Cartesian geometry axes.

MCS

The machine coordinate system (= MCS) refers to the coordinates of the machine axes, i.e. all machine axes are displayed in the machine coordinate system.

Machine position	Repos offset
X	
Y	
Z	

WCS

An offset (e.g. zero offset, rotation) can be used to set up a relationship, e.g. with the workpiece clamp. This relationship defines the position of the workpiece coordinate system (= WCS) in relation to the machine coordinate system. The workpiece is always represented in a Cartesian coordinate system.
All geometry axes and special axes are displayed in the workpiece coordinate system.

Work position	Repos offset
X1	
Y1	
Z1	

Machine manufacturer

Machine data are used to define whether the programmed frames are to be calculated when displaying the WCS (settable zero system = SZS display).
Please refer to the information and specifications provided by the machine manufacturer!

Operating sequence

"Jog" mode is selected in the "Machine" operating area.

Act. val. MCS

The actual values of the machine axes and their positions are displayed.
The softkey label changes to "Act. val. WCS".
The machine coordinate system comprises all the physically existing machine axes. References points, tool and pallet change points are defined in the MCS.

Act. val. WCS

When you select softkey "Act. val. WCS", the geometry and auxiliary axes plus their positions appear in the "Position" window.
The softkey label changes to "Act. val. MCS".

The workpiece coordinate system is assigned to a specific workpiece. Settings in the NC program refer to the WCS.

Zoom act. val.

Once the softkey has been pressed, the display comes up in enlarged view.
You can get back to normal view via the Recall key.

WCS MCS

- You can also toggle between the workpiece and machine coordinate systems by pressing the "WCS/MCS" key on the MCP.
- The number of places displayed after the decimal point and units of measure can be set in machine data.

Function

Coordinate system for changing the actual value display

Via the MD you can define whether the actual values are to be displayed

- without the programmed offsets = WCS or
- including programmed offsets = SZS (settable zero offset system).

Example:

Program	WCS display	SZS display
....		
N110 X100	100	100
N120 X0	0	0
N130 $P_PFRAME=CTRANS(X,10)	0	0
N140 X100	100	110
N150 ...		

References

/FB1/K2: Coordinate Systems…

4.2.4 Display axis feedrates

Function

In operating mode "Jog" or "MDA" or "AUTO", you can display the current feedrate, residual path information and the associated override data.

Operating sequence

"Jog" mode is selected in the "Machine" operating area.
Press the "Axis feedrate" softkey:

Axis feedrate

- On "MCS" the feedrate window with the current feedrates and the residual path information, as well as the associated override data, is displayed.
- On "WCS", the feedrate window is displayed for the axes that are subject to the interpolation, with current feedrate, residual path information and path override, as well as for the other axes with current feedrate, residual path information and single axis override.

You can use the "Page" keys to display other axes if required.

PAGE DOWN or PAGE UP

4.2.5 Display spindles

Function

The current spindle values (actual speed, setpoint speed, position on oriented spindle stop and spindle override) can be displayed.

Operating sequence

"AUTO"/"MDA"/"Jog" mode is selected in the "Machine" operating area.

Spindle

The "Spindle" window appears on the screen.
The window contains the spindle setpoint and actual value, position, setting of spindle override switch and spindle output.
The spindle load for 2 analog spindles in total can be displayed in the window.

PAGE DOWN or PAGE UP

Using the "Page" keys you can scroll up and down to display other spindles if any are configured.

Further information

- The "Spindle" window is displayed only if at least one spindle is configured.
- If a master spindle “S1” is configured, this is automatically displayed in the spindle window, even if it is not the first spindle.

4.2.6 Display help functions

Function

Auxiliary functions active in the selected channel can be displayed.

Operating sequence

"Auto"/"MDA"/"Jog" mode is selected in the "Machine" operating area.

Auxiliary functions

The "Auxiliary functions" window appears on the screen.
Up to 3 M functions or 3 H functions can be displayed.

References /PG/Programming Guide Fundamentals

4.2.7 Display G functions, transformations and swivel data record

Function

Active G functions and transformations can be displayed in the active channel.

Operating sequence

"AUTO"/"MDA"/"Jog" mode is selected in the "Machine" operating area.

Transform./ G functions

The window "Transform. + G functions" with the active G functions and transformations is displayed.

PAGE DOWN or PAGE UP

Using the "Page" keys you can scroll up and down to display other G functions.

Further information

Every G group has a fixed location.
The group number (no.) and the current G function of the G groups are displayed only if a G function is active.

References /PG/Programming Guide Fundamentals

Swivel data status display

Precondition:
The swivel function (CYCLE800) has been set up by the machine manufacturer.

The active swivel data record (orientable toolholder TOOLCARRIER) is displayed concurrently with the transformations.
The following values are displayed:
TCARR = 1 .. n* *n max. no. of set swivel data (TOOLCARRIER) that are active.

References /PGZ/Programming Guide Cycles: Chapter 3 "Swiveling"

4.2.8 Handwheel

Function

Using the "Handwheel" function, you can assign an axis to the handwheels and activate them.

Operating sequence

"Jog" mode is selected in the "Machine" operating area.

Handwheel

The "Handwheel" window appears on the screen.

Position the cursor on the handwheel of your choice (1–3).

An axis identifier is suggested in the "Axis" field. All other existing axes can be selected via the "Select" key. The settings are accepted immediately and an axis is assigned to each handwheel (1–3).

Every time you press the "Select" key in the "Active" field you activate or deactivate enabling of the relevant handwheel. The settings become active immediately.
When you turn the handwheel, the assigned axis traverses by the number of increments set for it ("Inc" keys).

Machine manufacturer

The machine-tool manufacturer is responsible for the design of handwheels. Operation may therefore differ from the explanation above.
Please refer to the information and specifications provided by the machine manufacturer.

4.2.9 Preset

Function

The "Preset" function can be used to redefine the control zero in the machine coordinate system.
The preset values act on machine axes.
Axes do not move when "Preset" is active.

Danger

After the actual value has been reset, none of the protection zones or software limit switches are operative! Only after another reference point approach are the protection zones and software limit switches active again.

Further information

A new position value is entered for the current axis positions.

Machine manufacturer

Please refer to the information and specifications provided by the machine manufacturer.

Operating sequence

"Jog" mode is selected in the "Machine" operating area.

Preset

The "Preset" window appears on the screen.

Enter the new actual value, which must in the future correspond to the current axis position for each individual axis. By doing so, you are redefining the control zero in the MCS. When the control zero is redefined, the tool change point, for example, also changes.

Further information

Machine manufacturer

The "Preset" function can be disabled by means of protection levels (keyswitch position).

4.2.10 Set actual value

Function

The "Set actual value" function sets the workpiece coordinate system to a defined actual coordinate and calculates the resultant offset between the old and a newly entered actual value in the WCS in the 1st basic offset.

The "Set actual value" function can only be used if the control is in the workpiece coordinate system.
The functions are stored under the same softkey.

Machine manufacturer

Please refer to the information and specifications provided by the machine manufacturer.

References

/IAM/ Installation & Start-Up Guide /IM2, HMI Embedded

Set actual value assigns the 1st base offset, rough.

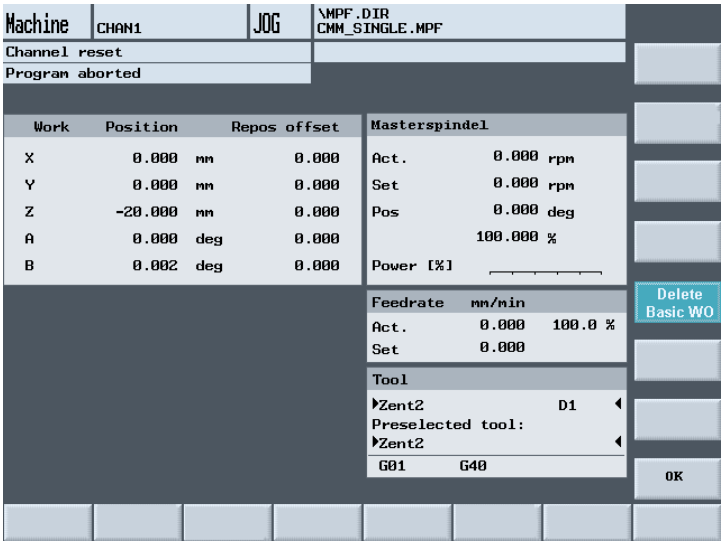
Operating sequence

Set actual value

1 ... 9

INPUT

The new setpoint position of the axes in the workpiece coordinate system can be entered using "Set actual value" in the actual value window. When you transfer a value to the system by pressing "Input", the deviation from the current actual value is entered in the basic offset. The new actual value is displayed in the "Position" column.



Delete Basic WO

OK

"Delete Basic WO" can be selected to undo all the previously entered offset settings.
Confirm with "OK".

4.2.11 Inch/metric switchover

Function

The control system can operate with the inch or the metric system of measurement. You can switch between the inch and metric measuring systems in the "Machine" operating area. The control converts the values accordingly.

Machine manufacturer

The switchover is only possible if:

- The corresponding machine data have been set.
- All channels are in the Reset state.
- Axes are not traversing with JOG, DRF or PLC control.
- Constant grinding wheel peripheral speed (GWPS) is not active.

The display resolution for the inch system of measurement is defined by machine data.

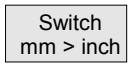
Actions such as parts program start or mode change are disabled for the duration of the switchover.

Operating sequence

"AUTO"/"MDA"/"Jog" mode is selected in the "Machine" operating area.

Press the ETC key followed by

the "Switch mm > inch" softkey.



When the system of measurement is changed, all length-related parameters are automatically converted to the new system of measurement from the perspective of the operator.

If the switchover cannot be performed, this is indicated by a message in the user interface.

References

/FB1/G2: Section: Metric/Inch System of Measurement

4.2.12 Actual value display: Settable zero offset system, SZS

Function

You can make a setting in the MD to define whether

- the position of the workpiece coordinate system, WCS (= programmed position, corresponds to default setting) or
- the tool holder position of the active tool relative to workpiece zero (settable zero offset system)

is to be displayed in the actual value display.

References

For configuring instructions see:
/IAM/IM2, Installation & Start-Up Guide HMI Embedded:
Chapter "Zero Offset"

4.3 Reference point approach

Function

The "Ref Point" function ensures that the control and machine are synchronized after power ON.
Before a reference point approach can be carried out, the axes must be located at positions (if necessary, traversed to these positions using the axis keys/handwheels) from where the machine reference point can be approached without collision.
If reference point approach is called from a part program all axes can be traversed simultaneously.

Reference point approach can only be performed by machine axes.
The actual value display does not match the real position of the axes when the control is switched on.

Caution

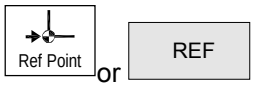
- If the axes are not safely positioned, you must traverse them to safe positions in "Jog" or "MDA" mode.
- You must follow the axis motions directly on the machine!
- Ignore the actual value display until the axes have been referenced!
- The software limit switches are not active!

Machine manufacturer

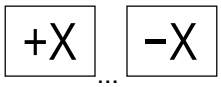
Please refer to the information and specifications provided by the machine manufacturer.

Operating sequence

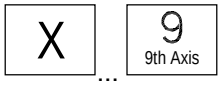
"Jog" or "MDA" is selected in the "Machine" operating area.
The channel for reference point approach is selected.



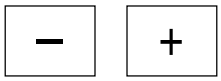
The "Ref Point" machine function is selected.



Turning machine:
Press the "Axis" keys.



Milling machine:
Select the axis to be traversed and



press the "+" or "–" key.

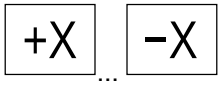
The selected axis moves to the reference point. The direction and sequence is defined by the machine-tool manufacturer in the PLC program.
If you have pressed the wrong direction key, the action is not accepted and the axes do not move.
The display shows the reference point value.

No symbol is displayed for axes that do not refer to a reference point.

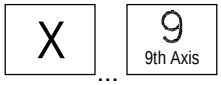
○ This symbol appears for axes that must be referenced.

◒ This symbol is displayed next to the axis when the reference point has been reached.

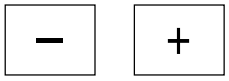
The axis, once started, can be stopped before it reaches the reference point.



Turning machine:
Press the "Axis" keys.



Milling machine:
Select the axis to be traversed and



press the "+" or "–" key.
The selected axis moves to the reference point.

Caution

The machine is synchronized as soon as the reference point is reached. The actual value display is set to the reference point value. The display is the difference between the machine zero and the slide reference point. From now on path limits, such as software limit switches, are active.

You can end the function by selecting another operating mode ("Jog", "MDA" or "Automatic").

- All axes of a mode group can approach the reference point simultaneously (depending on the PLC program of the machine-tool manufacturer).
- The feedrate override is active.

Machine manufacturer

Your machine manufacturer will instruct you how to select axes if you intend to enter more than nine.

Further information

The sequence in which axes must be referenced can be defined by the machine-tool manufacturer.

The machine can be started in Automatic mode only when all axes with a defined reference point (see machine data MD) have reached it.

4.4 Jog operating mode

4.4.1 Function and basic display

Function

In manual mode you can:

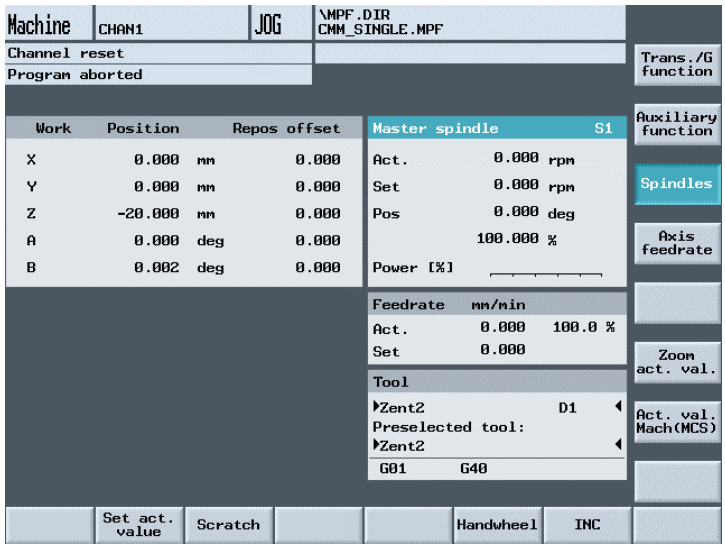
1. Synchronize the measuring system of the control with the machine (reference point procedure).
2. Set up the machine, i.e. activate manually controlled motions on the machine using the keys and handwheels provided on the machine control panel.
3. Activate manually controlled motions on the machine using the keys and handwheels provided on the machine control panel while a part program is interrupted.

or JOG

The following basic display "Jog" is displayed when you press the "Area switchover" key

and the "Jog" key.

The "Jog" basic display contains values relating to position, feedrate, spindle and tool.



Explanation of the Jog basic display

Motion symbol in actual value window: The axes are still moving, i.e. they are not within the exact stop window.

WCS
\+ X2*
Y2*
Z2*

Displays the addresses of the existing axes with the machine axis identifier (MCS) or with the geometric axis identifier (WCS).
(see also Section "Toggling between Machine/Workpiece Coordinate System (MCS/WCS)")

Machine manufacturer

Machine data are used to define whether the programmed frames are to be calculated when displaying the WCS (settable zero system = SZS display). Please refer to the information and specifications provided by the machine manufacturer.

- If the axis identifier can only be displayed in an abbreviated form it is replaced by the character *.
- If you traverse an axis in the positive (+) or negative (-) direction, a plus or minus sign is shown in the relevant field.
  The axis is in position if neither + nor – is displayed in the position display.

Position
0.0
0.1
–0.1
1.1
0.0

The actual position of each configured axis in the machine (MCS) or work (WCS) is displayed in these fields.
The sign is only displayed for negative values.

Repos. offset
0.0
0.1
–0.1
1.1
0.0

If the axes are traversed in the "Program interrupted" status in "Jog" mode, the path traversed by every axis with respect to the point of interruption is displayed in the REPOS offset.

Spindle
rpm

Spindle window (if spindle is available)
Displays the set and actual value of the spindle speed, the position of the spindle, the position of the spindle override switch  , and the spindle power.

Nibbling
The spindle window is replaced by a nibble window if the "Nibbling" technology option is set.
The active function, if any, appears at the top left in the window header:

- PON Punching ON
- SON Nibbling ON
- SPOF Punching/nibbling OFF

The following values are displayed:

- “Section” and “Number of strokes” are displayed inverted, if you have programmed “Length of section” or “Number of sections” at the automatic block partition.
- The “Time delay” is only displayed if you have set “Punching with time delay”.

Feed
mm/min

Feedrate window
Display of the setpoint and actual value of the feedrate as well as the position of the feedrate override switch  (in %). The actual setpoint to be traversed is dependent on the override switch.
When G00 (rapid traverse movement) is programmed, the rapid override value is displayed.

Tools

Tool window
Display of active tool offset (e.g. D1), tool currently in use (T no.), pre-selected tool (on milling machines) plus currently active travel commands (e.g. G01, SPLINE, ...), or
tool radius compensation not active (e.g. G40).

Zoom
act. val.

Increases the size of the actual-value display.

4.4.2 Traversing axes

Traversing speed

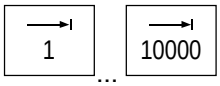
The initial settings for traversing velocity and feed mode are stored in setting data for JOG mode.
Traversing velocities are defined by the machine-tool manufacturer.
The default setting for the feedrate is mm/min.
See operating area "Parameters/Setting data/Jog data".

Operating sequence

"Jog" mode is selected in the "Machine" operating area.

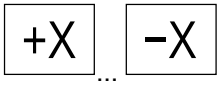
Traverse the axes
Using the "Inc" (increment) function, manually traverse the selected axis in preset increments in the appropriate direction by pressing an "Axis key" repeatedly:

- [.] Variable increment can be set via softkey "Inc" (see Subsection 3.2.5).



- [1], [10], ..., [10000] fixed increment

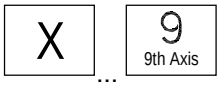
Turning machine:
Press the "Axis" keys.

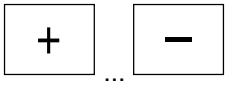


If necessary, set the speed with the override selector.

If you press "Rapid traverse override" at the same time, you can traverse the axis in rapid traverse mode.

Milling machine:
Select the axis to be traversed and





press the "+" or "–" key.

Feedrate and rapid traverse override switches can be operative.

One or several axes can be selected at the same time (depending on PLC program).

Further information

- When the control is switched on, axes can be traversed to the limits of the machine because they have not yet been referenced. Emergency limit switches might be triggered as a result.
- The software limit switches and the working area limitation are not yet operative!
- The feedrate enable must be set (FST display must not light up in the program control display).

Machine manufacturer If no advantage is to be gained from moving several axes simultaneously, the machine manufacturer must implement the appropriate interlock in the PLC program.

4.4.3 Inc: Incremental dimension

Function

The "INC" (Increment) function can be used to enter a settable increment value for Inc variable traversing.

Operating sequence

"Jog" mode is selected in the "Machine" operating area.

INC

The "Increment traverse keys" window appears on the screen.

Enter the desired “INC-Var” increment size

Abort

Go to the previous screen form again without accepting value.

OK

Click on “OK” to save.

→| VAR

Press this key in manual mode together with the "Axis" key to traverse the axis in increments of the size set above in the appropriate direction (see also Section "Traversing axes").
Increment keys with preset increment sizes are temporarily inoperative.

4.4.4 Repos

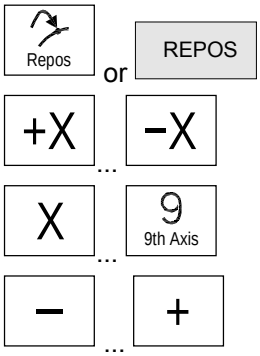
Function

After a program interruption in automatic mode (e.g. to take a measurement on the workpiece or to compensate tool wear values or after tool breakage), you can retract the tool manually from the contour after selecting "Jog" mode. In such cases, the control stores the coordinates of the point of interruption and displays the path distances traversed by the axes in "Jog" mode as a "Repos" offset (Repos = repositioning) in the actual value window.

"Repos" offsets can be displayed in the machine coordinate system (MCS) or workpiece coordinate system (WCS).

Operating sequence

"Jog" mode is selected in the "Machine" operating area.
The axes have been moved away from the point of interruption.



Select machine function "Repos".

Turning machine:
Press the "Axis" keys.

Milling machine:
Select the axis to be traversed and

press the "+" or "–" key.

It is not possible to pass over the point of interruption.
The feedrate override switch is active.

Warning
The rapid traverse override switch is active.
Non-adjusted Repos offsets are adjusted on switchover to Automatic mode followed by start with program advance and linear interpolation.

4.4.5 SI (Safety Integrated): User confirmation

Function

If the "User enabling" option is installed on the NC, you must enable or disable the function depending on the keyswitch position in the "Approach reference point" operating mode.

User enabling can only be granted if keyswitch position 3 or higher is selected for the keyswitch position.

The displayed values always refer to the machine coordinate system (MCS).

Operating sequence

"Jog Ref" mode is selected in the "Machine" operating area.
The channel to be referenced is already selected.

User confirmation

Press softkey "User confirmation".
The "Confirm machine positions" window is opened.
The machine axes in the MCS are displayed, together with the current position and a checkbox for activating/deactivating the user confirmation.

Place the cursor on the required machine axis.

SELECT

Activate or deactivate the user confirmation for the selected machine axis via the "Select" key.

Check whether a reference point approach has been performed on the relevant axis. If not, error message "Please reference axis first" is displayed. User agreement cannot be activated for the axis until it has been referenced.

Further information

The function user enabling is only provided if user enabling is required for at least one axis of the channel.

References

For more information please refer to:
/FBSI/, Description of Functions SINUMERIK Safety Integrated.

4.4.6 Determine scratching/zero offset

Function

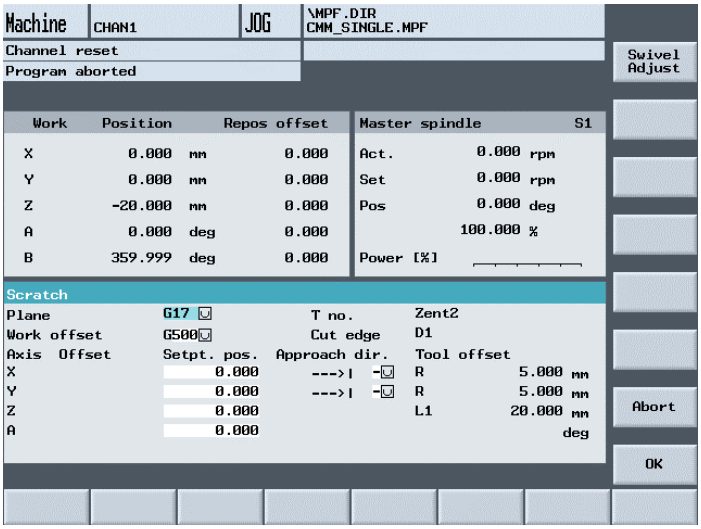
You can determine the zero offset by "scratching", taking an (active) tool and, if necessary, the basic offset into account.
A window is provided for the "Scratching" function.

Scratch

Operating sequence

1. Press softkey "Scratch":

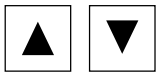
- The active plane is displayed and can be altered (via "Select" key).
- The active WO is displayed and can be altered (via "Select" key).
- The active tool is displayed. No tool is displayed if none is active (message).



Meanings of columns in the "Scratch" window:

- "Offset": Current value of offset to be determined. The coarse offset is displayed. The fine offset is taken into account and remains valid.
- "Set position": Input of subsequent setpoint position for scratched edge.

"Approach direction": Selection fields for positive/negative approach direction.



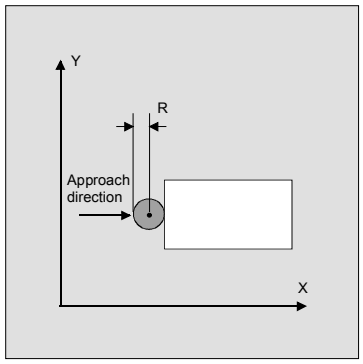
Use the cursor to select the first axis to be moved in the "Scratch" display.

2. Move the axis up to the workpiece, enter the chosen setpoint position (e.g. "0") and press the "Input" key. The offset is then calculated.
   Repeat the process for other axes.

3. Press "OK" to transfer all values to the selected WO. The offset is always calculated in relation to the current work (WCS).

4. To change the direction of approach, position the cursor on the axis to be modified in the "Approach direction" column and press the "Select" key.

Example:



Set "+R" using the "Select" key.

The values defined by scratching are displayed in the basic screen "Overview of WO" under system frame "Set zero position" if G500 was selected as work offset. Otherwise, the data are saved in the "selected settable WO".

Machine manufacturer

Please refer to the information and specifications provided by the machine manufacturer!

Scratching in swiveled plane (NC software version 6.3 and higher)

Precondition:
The swivel function (CYCLE800) has been set up by the machine manufacturer.
The "Activate swivel data" softkey is only displayed if a TOOLCARRIER is active.

The "Swivel Adjust" softkey is used to declare the swiveled plane as the new "zero" plane for the active swivel data record (TOOLCARRIER).
In this case, when you select "Swivel", positioning is at this swiveled plane with rotation through
X=0
Y=0
Z=0
The following messages are output for checking purposes:
"Swivel: Adjustment terminated"
"Swivel: Adjustment not possible"

References /PGZ/ Programming Guide Cycles, Chapter 3 "Swiveling"

4.4.7 Displaying system frames

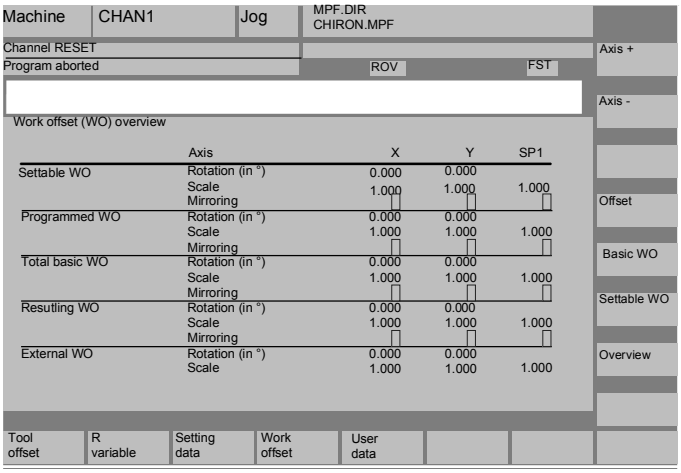
If system frames are activated via MD, they can be displayed via the Parameter operating area, Work offsets.

The following assignment is applicable:

$P_SETFR	System frame for actual value setting, scratching
$P_EXTFR	System frame for work offset external
$P_PARTFR	System frame for TCARR and PAROT
$P_TOOLFR	System frame for TOROT and TOFRAME
$P_WPFR	System frame for workpiece reference points
$P_CYCFR	System frame for cycles

Both the offset set via frames (coarse and fine) and the rotation and mirroring defined there are displayed. Display is according to the position in the frame chain.

The following figure shows an example screen layout:



Machine manufacturer Please refer to the information and specifications provided by the machine manufacturer.

4.5 MDA operating mode

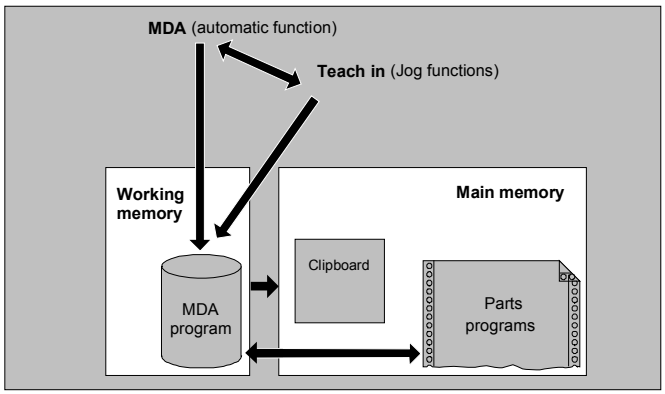
4.5.1 Function and basic display

In "MDA" (Manual Data Automatic) mode, you can write part programs block by block and execute them. You can transfer the required motions as single part program blocks to the control using the operator panel.
The control starts processing the entered blocks when you press the "Cycle Start" key.

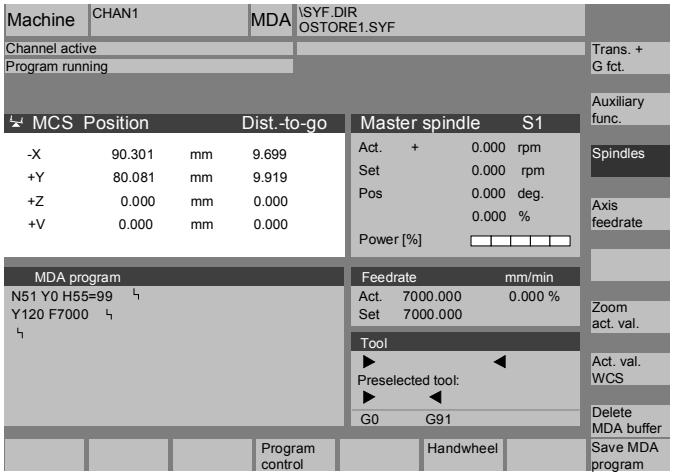
Caution
The same safety interlocks must be applied as used in fully automatic operation. The same preconditions must be fulfilled as for fully automatic operation.
The automatic functions (traverse blocks) are active in "MDA" mode.

Teach In

The functions associated with "Jog" are active in submode "Teach In" and can be accessed via an MCP key. You can therefore create and store a program in the input and manual modes by alternating between "MDA" and "Teach In".
You can use the editor to edit the program blocks in the MDA window. You can view blocks that have already been processed by paging upwards.
Blocks that have already been executed can only be edited in the Reset state.
Further blocks can be added with "Input".
Blocks can be input and operated only with the channel in the "Channel Reset" or "Channel interrupted" state.
The program created in MDI mode is saved in the "MPF" directory as a part program (MPF).



The "MDA" basic display contains values relating to position, feedrate, spindle and tool as well as the contents of the MDA clipboard.



Explanation of "MDA" basic display

Analogous to the Jog basic display, the actual value window, spindle window, feedrate window and tool window are output.

Save MDA program

The "Save MDA program" softkey stores the MDA program in the "MPF" directory.

Vertical softkeys

Like the Jog basic display, the MDA contains the "Trans. + G fct." "Auxiliary functions" and "Spindles" softkeys (if spindle is configured).

Delete MDA prog.

The contents of the MDA program in the NC are erased.

4.5.2 Save program

Function

A program created in MDI can be temporarily stored in directory "MPF" and permanently

- incorporated as a program or
- read out to an external device via the RS-232 C interface.

Save MDA program

The system will ask you to enter a name for the file to be saved to the MDA program.
The program is saved/stored as a part program (MPF.dir) under the specified name in directory "MPF".

4.5.3 Teach In

Function

With the "Teach In" function, part programs (main programs and subroutines) for motion sequences or simple workpieces can be created, modified and executed by approaching and then storing positions in combination with the function "MDA".

There are two possible methods of writing programs with "Teach In" and "MDA":

- Manual positioning.
- Manually inputting the coordinates and additional information.

Both entry of blocks (input, delete, insert) and automatic insertion by manual approach of positions are possible only at points that have not yet been executed.

1. Manual positioning

Operating sequence

"MDA" mode is selected in the "Machine" operating area.
The program is not running.

Submode "Teach In" is selected.
After selecting the function, the cursor is located in the first blank line of the "Teach In program" window.

+X ... −X

Turning machine:
Press the "Axis" keys.

X ... 9 (9th Axis)

Milling machine:
Select the axis to be traversed

\+ −

and then press the "+" or "–" key.

The axis name and the path being traversed are displayed continuously on the screen in the "Teach In program" window.
The axis name and axis positions are transferred to the MDA program as values referred to the workpiece coordinate system.

Cycle Start

Saving the position value:
The position values of the axes can be altered in the clipboard until this block is saved with the "Cycle Start" key.

When you use the "Save block" function for the first time you will be asked to enter the name of the teach program.
The teach program is now set for the duration of the MDA/Teach in procedure.
After "Jog" or "AUTO" mode has been selected, a new Teach program can be set.

Additional functions:
Place the cursor at the desired position in the "Buffer" window.

Enter the additional functions (e.g. feedrates, miscellaneous functions, etc.) in the program (if permitted).

Delete/insert/save block:
Position the cursor at the desired point.

Delete block

The block is deleted automatically.

Insert block

The block stored in the delete memory is automatically inserted in front of the line in which the cursor is located.

Save block

New position values and miscellaneous functions are saved.

When "Cycle Start" is pressed, the appropriate travel motions and functions additionally entered are executed as program blocks. While the program is being processed, the blocks traversed by the NC are displayed in the "Current block" window.

2. Manual input of coordinates

Operating sequence

"MDA" mode is selected in the "Machine" operating area.

Save position values/additional functions:
Transfer the coordinates of the traversing positions plus any additional functions (preparatory functions, auxiliary functions, etc.) in the program by entering them in the "MDA program" window.

Further information

- Changes to the zero offset cause the axis to execute compensatory motions after Cycle Start.
- In the case of G64, the end point response will differ when the part program is executed in "Automatic" mode.
- All the G functions can be used.

4.6 Automatic mode

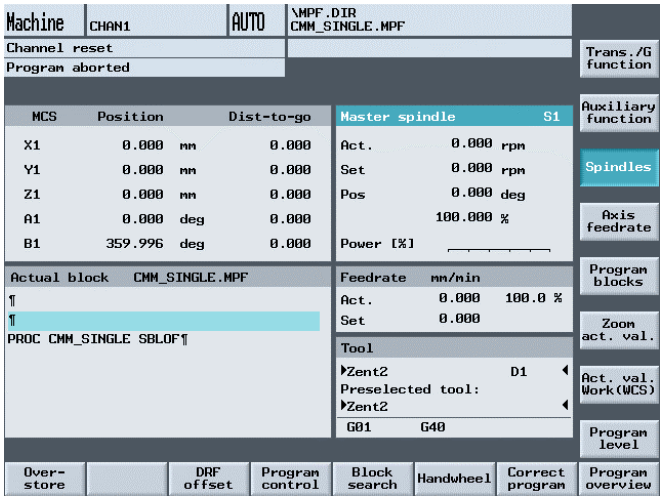
4.6.1 Function and basic display

Prerequisites

You can execute part programs fully automatically in "Automatic" mode; this is the normal operating mode for part machining.
The following conditions must be fulfilled before you can execute part programs:

- You have already synchronized the control measuring system with the machine (i.e. "approached" reference points).
- You have already loaded the associated part program in the control.
- You have checked or entered the necessary offset values, such as work offsets or tool offsets.
- The required safety interlocks are already active.

The "Automatic" basic display contains values relating to position, feedrate, spindle, and tool as well as the block currently being processed or program pointer.



Explanation of basic display

Like the Jog display, the Automatic display contains actual value, spindle window, feedrate and tool windows.

If the NC detects G0 during part program execution, the current value of the rapid traverse override is displayed in the "Feedrate" window.

Horizontal softkeys

Program overview

The workpiece or program overview is displayed. In this area it is possible to select programs for execution.

Vertical softkeys

Current block

The "Current block" window is also displayed.
The current block is highlighted while the program is running.
The name of the program to which the blocks on the screen belong is output in the window header.

Program level

When you press the "Program level" softkey, the window headed "Program level" appears instead of the "Current block" window.
The program nesting depth (P = number of passes) is displayed.

Program level or Current block

In program operation it is possible to toggle between the "Program level" and "Current block" displays.

Program blocks

Seven program blocks of the current program are displayed together with the current position in the part program.
The 7-block display always shows the programmed part program and not the actual program run. In particular with subroutine calls, the block displayed as next block after the UP is only the block which is executed after the UP call, not the first block of the subprogam (program execution).

Further information

The other softkeys are described in the sections below.

4.6.2 Execute program

Function

After selection of a workpiece or program overview, individual workpieces or programs can be enabled or disabled for execution.

Operating sequence

The "Auto" operating mode is selected in the "Machine" operating area.
The appropriate channel is selected.
The channel is in reset state.
The workpiece/program to be selected is in the memory.

Program overview

When you press softkey "Program overview", the software automatically changes to the "Program" operating area. An overview of all workpiece directories/programs is displayed.
See Chap. 6:

- Execute a program on the NC for execution
- Execute program from network drive, compact flash card, diskette.

Once you have selected the desired workpiece/program for execution, use the machine area key to change back to the "Machine" operating area.

The program starts running when you press the "NC Start" key, and you can track the axis position, spindle, etc in the display.

4.6.3 Program editing

Function

As soon as a syntax error in the part program is detected by the control, program processing is interrupted and the syntax error is displayed in the alarm line.
You can make small changes in the stop or reset state.

Operating sequence

"AUTO" mode is selected in the "Machine" operating area.

It is not possible to pass over the point of interruption. The feedrate correction switch is active.
The program status is in the "Stop" or "Reset" state.

Correct program

The correction editor is displayed with this softkey. If an error occurs, the faulty block is highlighted and can be corrected. The cursor is positioned on the error.

Cycle Start

Once you have corrected the block with an error, you can continue the program with the "Cycle Start" key.

- Stop state:
  Only program lines that have not yet been executed can be edited.
- Reset status:
  All program lines can be edited.
- Program Running state:
  Program cannot be edited!

4.6.4 Setting block search/search destination

Function

The block search function allows you to run the part program forward until you reach the block you require. Three types of search are available:

1. With calculation on contour:
   During block searches with calculation, the same calculations are performed as in normal program mode. The complete destination block is then executed analogously to normal program execution.

2. With calculation at block end point:
   During block searches with calculation, the same calculations are performed as in normal program mode. The interpolation mode valid in the destination block is then applied to approach its end point or the next programmed position.

3. Without calculation:
   No calculations are performed during the block search.
   The values stored in the control remain the same as they were before the block search.

You can define the search destination

- by direct positioning or
- by specifying a block number, a label, a string, a program name or any character string.

References

For further information about block search, please refer to:
/FB/ K1, Mode Group, Channel, Program control

Operating sequence

"Auto" is selected in the "Machine" operating area.
The channel is in the reset state.
The program in which the block search is to be performed is selected.

Block search

Calls up the "Search position" function.
Position the cursor on the destination block.
The block search is started when you press one of the following three softkeys:

Calculate contour

Block search start with calculation on contour

- When you press "Cycle Start", the axes execute a compensatory motion between the current actual position and the position of the block located by the block search.
- The new position is determined by the program status (all axis positions, active auxiliary functions) at the beginning of the selected block, i.e. after the block search, the control is positioned at the end position of the last NC block before the search destination.

Calc. block end point

Block search start with calculation at block end point

without calculation

Block search start without calculation

- If the destination block is found this block becomes the current block. HMI Embedded signals "Search target found" and displays the target block in the current block display.
- Alarm 10208 is output to indicate that operator interventions such as Overstore or Mode Change after JOG are permissible.
- After "Cycle Start" the program is started and executed from the target block.

Reset

Block search can be aborted with Reset.

Define search destination in program editor:
The current selected program level is displayed.

Search position

Position the cursor bar on a destination block of your choice in the part program.

Program level + or Program level –

If the program interruption occurred in a subprogram level, you can change program levels here.

Search pointer

Defining the search destination in the search pointer:
Once the softkey "search pointer" has been pressed, a screen form with the program pointer is displayed.

It contains input fields for program name, search type (block number, text, ...) and search destination (content). The cursor is positioned in the input field for the "search type."

0 ... 5

You must enter your selected search type for the search destination in the input field "Type."

Information about the available search types is displayed in the dialog line.

The following search types can be entered:

Search type (= jump to ...)	**Value in search type field**
Program end	0
Block number	1
Jump label	2
Any character string	3
Program name	4
Line number	5

Different search types can be specified for different program levels.

A ... Z

You can enter your chosen search destination (according to search type) in the "Search destination" field.

0 ... 9

You can enter the corresponding number of program passes in the "P" field (pass counter).

Interrupt. point

Preset search destination to last program interruption point:
The search pointer is assigned the data of the last program interruption point.

4.6.5 Accelerated block search for execution from external source

Function

The block search function allows you to run the part program forward until you reach the block you require.
In the "Search position" and "Search pointer" menus, you can use the "External w/o calc." softkey to start an accelerated block search on programs executed from an external device.
You can define the search destination

- by directly positioning the cursor on the target block, or
- by specifying a block number or a line number.

Block search sequence:
In certain circumstances, external program parts are not transferred or are only partially transferred to the NC.
Only those programs and program parts which are needed in order to reach the specified search target and to continue program execution are transferred to the NC. This applies to the following:

- "Execution from external source" function
- Execution of EXTCALL instructions.

Notice
Since certain program parts may not have been transferred to the NC, it must be assumed that modal functions such as feed and motion commands are incorrect at the target block (main block).
Consequently, when using the variants "Without calculation" and "External – without calculation", you will either need to make sure that any information required for machining is available in the selected target block (main block) and following blocks, or use the override to complete the necessary settings.

Operating sequence

"Auto" is selected in the "Machine" operating area.
The channel is in the reset state.
The program in which the block search is to be performed is selected.

Block search

Switch to the "Search position" dialog.

Define search destination in program editor:

Search position

The current selected program level is displayed.
The block search via "Search position" is only possible for programs which have been loaded onto the NC and not for programs executed from the hard disk.

Position the cursor bar on a destination block of your choice in the part program.

Program level + or Program level –

If an interruption point is available on the NC, you can switch back and forth between the program levels.

External w/o calc.

Start search without calculation for external programs.

Defining the search destination in the search pointer:

Search pointer

Once the softkey "Search pointer" has been pressed, a screen form with the program pointer is displayed.
It contains input fields for program name, search type (block number, text, ...) and search destination (content). The cursor is positioned in the input field for the "search type".
Type 1 (block number) and type 5 (line number) are possible as search target types.

External w/o calc.

Start search without calculation for external programs.

4.6.6 Overstoring

Function

In "Auto" mode you can overstore technological parameters (auxiliary functions, all programmable instructions ...) in the working memory of the NC. You can also enter and execute any NC block.

Operating sequence

"Auto" is selected in the "Machine" operating area.

Cycle Stop

Stop the program by selecting "Cycle Stop."

Overstore

The "Overstore" window is opened.

In this window you can now enter the desired NC blocks that are to be processed.

Cycle Start

On "Cycle Start" the entered blocks are executed, the "Current block" window displayed and the softkeys hidden. The "Overstore" window and associated softkeys are not displayed again until the "Channel interrupted", "Stop" or "Reset" state is reached.

More blocks can be added (to the overstore buffer) when these blocks have been processed.

Note:

- After "Overstore" a subroutine with the content REPOSA is executed. The program is displayed for the operator.
- Overstoring is not possible in the "Program running" state.

Further information

- You can close the window and exit the Overstore function by pressing the "Recall" key. You have now exited the Overstore function.
- You cannot change operating modes until you have deselected "Overstore" with the "Recall" key.
- In "AUTO" mode the program selected prior to the overstore operation is now executed when you press "Cycle Start" again.
- Overstore does not alter the programs stored in the part program memory.
- The function is also available in single-block mode.

4.6.7 Program control

Function

You can use this function to change a program sequence in "Auto" and "MDA" modes. The following program control functions can be activated or deactivated:

SKP	Skip block
DRY	Dry run feedrate
ROV	Rapid traverse override
M01	Programmed stop
M101	Halt at cycle end (optional)
SBL1	Single block with stop after machine function blocks
SBL2	Single block with stop after each block.
DRF	Select DRF offset
PRT	Program test
-	Display all blocks in the current block display
-	Display only traversing blocks in the current block display window

See Chapter 2, “Program Control Display”

The display is dependent on the machine data of the operator panel front in which access authorization can be set.

References

/FB/, A2, Various Interface Signals: Chapter 4, or
/FB/ K1, Mode Groups, Channels, Program Operation

Operating sequence

"Auto" or "MDA" mode is selected in the "Machine" operating area.

Program control

The "Program control" window appears on the screen.

Place the cursor at the required position.

SELECT

Every time you press the "Select" key you activate or deactivate the selected function.

4.6.8 DRF offset

Function

The DRF (Differential Resolve Function) offset allows an additional incremental zero offset to be set using the handwheel. “DRF” is an axial traverse that works in the basic coordinate system. The handwheel selection is only permitted via the axial interface, i.e., only machine axes can be used.

Activation/deactivation

The DRF offset can be switched on and off for specific channels by means of the "Program control" function.
It remains stored until

- Power On for all axes
- DRFOF (deselect DRF by means of part program
- PRESETON (changing the actual value via Preset)

Change DRF

You can alter the DRF offset by traversing the appropriate machine axis using the handwheel (the actual value display does not change).

Operating sequence

"AUTO" mode is selected in the "Machine" operating area.
The standard axis assignment is defined.

Handwheel

Enter the desired handwheel or select via the MCP.

INC or [VAR]

Enter the desired increment or select via the MCP.

DRF offset

The window "DRF offset" is displayed.

Traverse the required axes using the handwheel.

Using the same operating sequence, you can also return the DRF offset to the value "0".

■

Parameters Operating Area

5.1 Tool data

5.1.1 Tool offset structure

A tool is selected in the program with the T function. Numbers T0 to T32000 can be assigned to tools. Each tool can have up to 9 cutting edges: D1 - D9. D1 to D9 activates the tool offset of a cutting edge for the active tool.
The tool length compensation is applied with the first traversing motion (linear or polynomial interpolation) of the axis.
A tool radius compensation is activated by programming G41/42 in the active plane (G17, 18, 19) and in a program block with G0 or G1.

Tool wear

Allowances for changes in the active tool shape can be made in the tool length (tool parameters 12-14) and tool radius (tool parameters 15-20).

Tool offset with D numbers only

Tool management is implemented outside the NCK. T numbers are irrelevant. This function is activated via the MD.
The D number range increases to 1 - 32000. A D number can be assigned only once for each tool, i.e. each D number represents precisely one tool offset data record.

Tool cutting edges

Each tool can have up to 9 cutting edges:
D1–D9.

5.1.2 Tool types and tool parameters

Entries

T No. Number of the tool
D No. Number of the cutting edge
Every data field (offset memory) that can be called with a D number contains not only the geometric information for the tool but also additional entries, i.e. the tool type (drill, milling cutter, turning tools with cutting edge position etc.).

Tool types

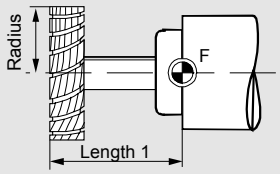
Tool type classification:

- **Group with type 1xy (cutter):**
  - 100 Cutter to CLDATA
  - 110 Ball end cylindrical die mill
  - 120 End milling cutter without corner rounding
  - 121 End mill with corner rounding
  - 130 Angle head cutter without corner rounding
  - 131 Angle head mill with corner rounding
  - 140 Facing tool
  - 145 Thread cutter
  - 150 Side mill
  - 155 Bevel cutter without rounding
  - 156 Bevel cutter with rounding
  - 157 Tapered die mill

Required offset values for a milling cutter

Entries in tool parameters

DP1	1xy
DP3	Length 1
DP6	Radius



Wear values according to requirement

Other values must be set to zero

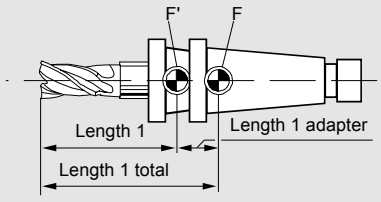
Effect	
G17:	Length 1 in Z Radius in X/Y
G18:	Length 1 in Y Radius in Z/X
G19:	Length1 in X Radius in Y/Z

F: Toolholder reference point

Milling tool with adapter

Entries in tool parameters

DP1	1xy
DP3	Length 1 -Geometry
DP6	Radius -Geometry
DP21	Length -Adapter



F: Adapter reference point (when tool is inserted = toolholder reference point)

Wear values according to requirements

Other values must be set to zero

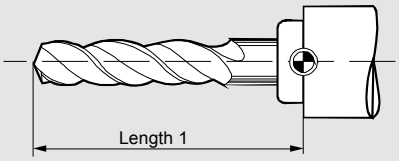
Effect	
G17:	Length 1 in Z Radius in X/Y
G18:	Length 1 in Y Radius in Z/X
G19:	Length 1 in X Radius in Y/Z

F´: Toolholder reference point

- **Group type 2xy (drills):**
  - 200 Twist drill
  - 205 Drill
  - 210 Boring bar
  - 220 Center drill
  - 230 Countersink
  - 231 Counterbore
  - 240 Tap regular thread
  - 241 Tap fine thread
  - 242 Tap Whitworth thread
  - 250 Reamer

Required offset values for a drill

Entries in tool parameters	
DP1	2xy
DP3	Length 1



Wear values according to requirements

Other values must be set to zero

Effect	
G17:	Length 1 in Z
G18:	Length 1 in Y
G19:	Length 1 in X

F: Toolholder reference point

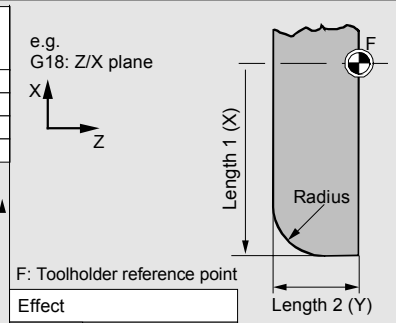
- **Group type 4xy (grinding tools):**
  - 400 Surface grinding wheel
  - 401 Surface grinding wheel with monitoring
  - 403 Surface grinding wheel with monitoring without base dimension for GWPS grinding speed
  - 410 Facing wheel
  - 411 Facing wheel with monitoring
  - 413 Facing wheel with monitoring without base dimension for GWPS grinding speed
  - 490 Dresser

Offset values required by a surface grinding wheel

Entries in tool parameters	
STC_DP1	403
STC_DP3	Length 1
STC_DP4	Length 2
STC_DP6	Radius

Wear values according to requirements

Other values must be set to zero



F: Toolholder reference point

Effect	
G17:	Length 1 in Y Length 2 in X Radius in X/Y
G18:	Length 1 in X Length 2 in Z Radius in Z/X
G19:	Length 1 in Z Length 2 in Y Radius in Y/Z

Required offset values for inclined grinding wheel with implicit monitoring selection

Entries in tool parameters	
STC_DP1	403
STC_DP3	Length 1
STC_DP4	Length 2
STC_DP6	Radius

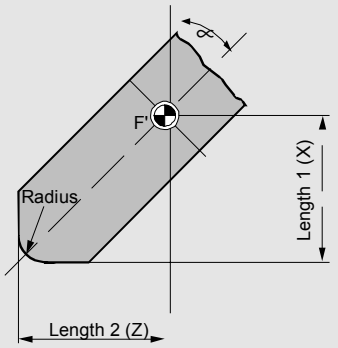
Wear values according to requirement

Other values must be set to zero

STC_TPG1	Spindle number
STC_TPG2	Chaining rule
STC_TPG3	Minimum wheel radius
STC_TPG4	Minimum wheel width
STC_TPG5	Current wheel width
STC_TPG6	Maximum speed
STC_TPG7	Max. surface speed
STC_TPG8	Angle of the inclined wheel
STC_TPG9	Parameter no. for radius calculation

F: Toolholder reference point

Effect	
G17:	Length 1 in Y Length 2 in X Radius in X/Y
G18:	Length 1 in X Length 2 in Z Radius in Z/X
G19:	Length 1 in Z Length 2 in Y Radius in Y/Z



Example of required offset values for inclined grinding wheel with implicit monitoring selection

Entries in tool parameters	
STC_DP1	403
STC_DP3	Length 1
STC_DP4	Length 2
STC_DP6	Radius

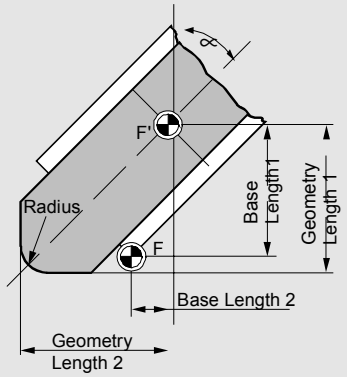
STC_TPG1	Spindle number
STC_TPG2	Chaining rule
STC_TPG3	Minimum wheel radius
STC_TPG4	Minimum wheel width
STC_TPG5	Current wheel width
STC_TPG6	Maximum speed
STC_TPG7	Max. surface speed
STC_TPG8	Angle of the inclined wheel
STC_TPG9	Parameter no. for radius calculation

Wear values according to requirement

Other values must be set to zero

Effect	
G17:	Length 1 in Y Length 2 in X Radius in X/Y
G18:	Length 1 in X Length 2 in Z Radius in Z/X
G19:	Length 1 in Z Length 2 in Y Radius in Y/Z

F: Toolholder reference point



Required offset values of a surface grinding wheel without base dimension for GWPS

Entries in tool parameters	
STC_DP1	403
STC_DP3	Length 1
STC_DP4	Length 2
STC_DP6	Radius
STC_DP21	L1 base
STC_DP22	L2 base

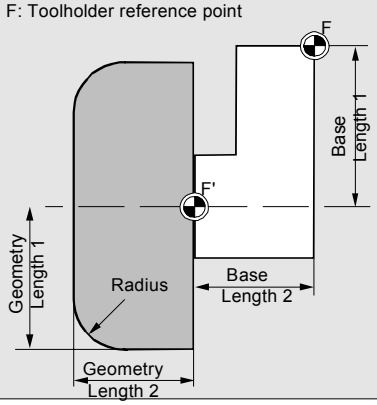
STC_TPG1	Spindle number
STC_TPG2	Chaining rule
STC_TPG3	Minimum wheel radius
STC_TPG4	Minimum wheel width
STC_TPG5	Current wheel width
STC_TPG6	Maximum speed
STC_TPG7	Max. surface speed
STC_TPG8	Angle of the inclined wheel
STC_TPG9	Parameter no. for radius calculation

Wear values according to requirement

Other values must be set to zero

Effect	
G17:	Length 1 in Y Length 2 in X Radius in X/Y
G18:	Length 1 in X Length 2 in Z Radius in Z/X
G19:	Length 1 in Z Length2 in Y Radius in Y/Z

F: Toolholder reference point



Required offset values of a facing wheel with monitoring parameters

Entries in tool parameters	
STC_DP1	403
STC_DP3	Length 1
STC_DP4	Length 2
STC_DP6	Radius

Wear values according to requirement

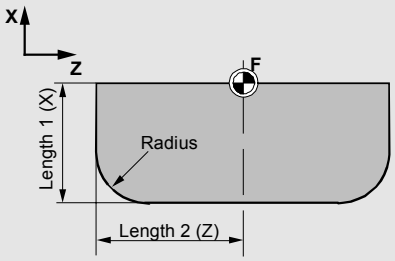
Other values must be set to zero

Effect	
G17:	Length 1 in Y Length 2 in X Radius in X/Y
G18:	Length 1 in X Length 2 in Z Radius in Z/X
G19:	Length 1 in Z Length 2 in Y Radius in Y/Z

STC_TPG1	Spindle number
STC_TPG2	Chaining rule
STC_TPG3	Minimum wheel radius
STC_TPG4	Minimum wheel width
STC_TPG5	Current wheel width
STC_TPG6	Maximum speed
STC_TPG7	Max. surface speed
STC_TPG8	Angle of the inclined wheel
STC_TPG9	Parameter no. for radius calculation

F: Toolholder reference point

e.g.
G18: Z/X plane

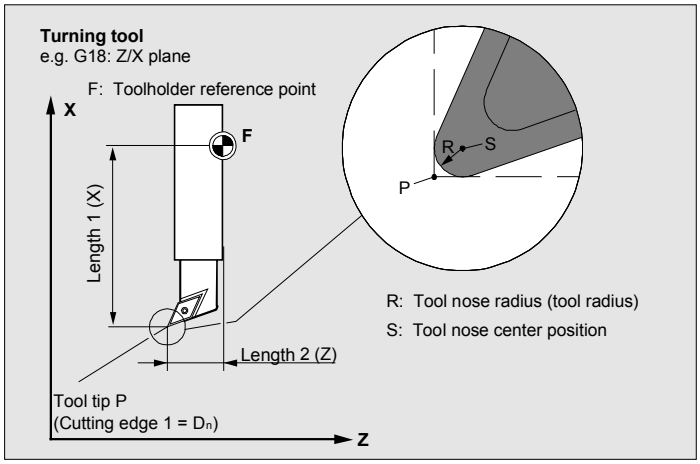


Assignment of Tool-specific Parameters

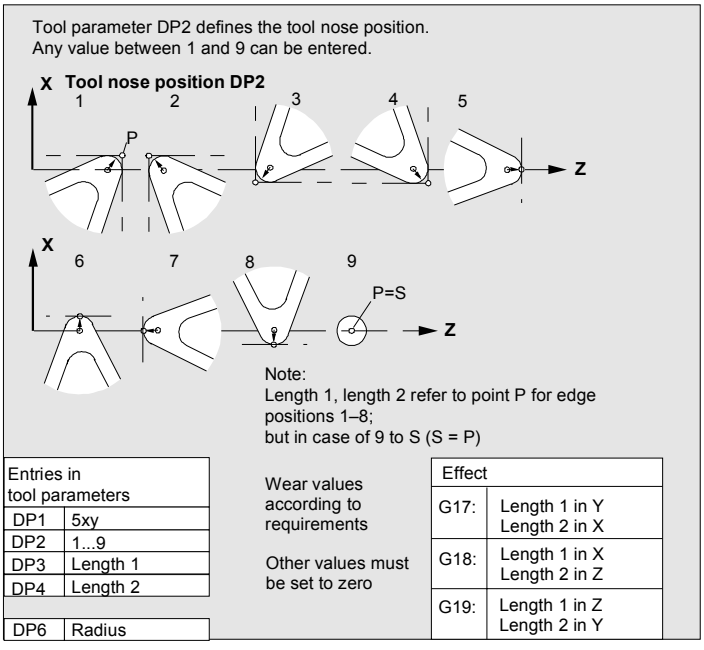
Parameters	Meaning	Data Type
Tool-specific parameters		
$TC_TPG1	Spindle number	Integer
$TC_TPG2	Chaining rule	Integer
$TC_TPG3	Minimum wheel radius	Real
$TC_TPG4	Minimum wheel width	Real
$TC_TPG5	Current wheel width	Real
$TC_TPG6	Maximum speed	Real
$TC_TPG7	Maximum surface speed	Real
$TC_TPG8	Angle of inclined wheel	Real
$TC_TPG9	Param. no. for radius calc.	Integer
Additional parameters		
$TC_TPC1	Angle of inclined wheel	Real
up to		
$TC_TPC10		Real

- **Group type 5xy (turning tools):**
  - 500 Roughing tool
  - 510 Finishing tool
  - 520 Plunge cutter
  - 530 Parting tool
  - 540 Threading tool

Required offset values for a turning tool with tool radius compensation



Required offset values for a turning tool with tool radius compensation



Entries in tool parameters	
DP1	5xy
DP2	1...9
DP3	Length 1
DP4	Length 2
DP6	Radius

Wear values according to requirements

Other values must be set to zero

Effect	
G17:	Length 1 in Y Length 2 in X
G18:	Length 1 in X Length 2 in Z
G19:	Length 1 in Z Length 2 in Y

- **Group type 7xy special tools**
  - 700 Slotting saw
  - 710 3D Probe
  - 711 Edge probe

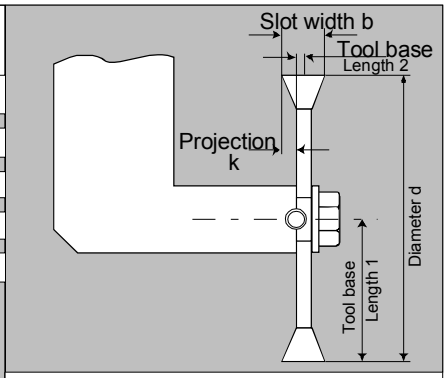
Example of required offset values for a slotting saw

Entries in tool parameters	
DP3	Length 1 - Base
DP4	DP3 Length 2 - Base
DP6	Diameter - Geometry
DP7	Zero width - Geometry
DP8	Projection - Geometry

Wear values according to requirement

Other values must be set to zero

L1 = DP3 + DP6/2
L2 = DP4 + DP7/2 - DP8
R = DP7/2



Effect		
G17	Semi-diameter (L1) in Z Projection in (L2) Y Saw blade in (R) X/Y	Plane selection 1st–2nd axis (X–Y)
G18	Semi-diameter (L1) in Y Projection in (L2) X Saw blade in (R) Z/X	Plane selection 1st–3rd axis (X–Z)
G19	Semi-diameter (L1) in Z Projection in (L2) Z Saw blade in (R) Y/Z	Plane selection 2nd–3rd axis (Y–Z)

The offset data (TOA data) you can enter for tool type 700 "slotting saw" are as follows:

	Geometry	**Wear**	**Base**	
Length compensation				
Length 1	$TC_DP3	$TC_DP12	$TC_DP21	mm
Length 2	$TC_DP4	$TC_DP13	$TC_DP22	mm
Length 3	$TC_DP5	$TC_DP14	$TC_DP23	mm
Radius compensation				
Diameter	$TC_DP6	$TC_DP15		mm
Slot width b	$TC_DP7	$TC_DP16		mm
Projection k	$TC_DP8	$TC_DP17		mm

Calculating the tool parameters

Types 1xy (milling cutters), 2xy (drills), and 5xy (turning tools) are calculated according to the same scheme.

Several entries exist for the geometric variables (e.g. length 1 or radius). These are added together to produce a value (e.g. total length 1, total radius) which is then used for the calculations.

Tool parameter number (P)	Meaning	Remarks
1	Tool type	For overview see list
2	Length of cutting edge	only for turning tools
Geometry	Length compensation	
3	Length 1	Calculation according to type and plane
4	Length 2	
5	Length 3	
Geometry	Radius	
6	Radius	Does not apply to drills
7	Reserved	
8	Reserved	
9	Reserved	
10	Reserved	
11	Reserved	
Wear	Length and radius compensation Radius compensation	
12	Length 1	
13	Length 2	
14	Length 3	
15	Radius	
16	Reserved	
17	Reserved	
18	Reserved	
19	Reserved	
20	Reserved	
Base dimensions/ adapter	Length offsets	
21	Length 1	
22	Length 2	
23	Length 3	
Technology		
24	Clearance angle	for turning tools
25	Clearance angle	

Offsets that are not required must be set to zero (= default when the offset memory is set up).
The individual values of the offset memory (P1 to P25) can be read and written by the program via system variables.

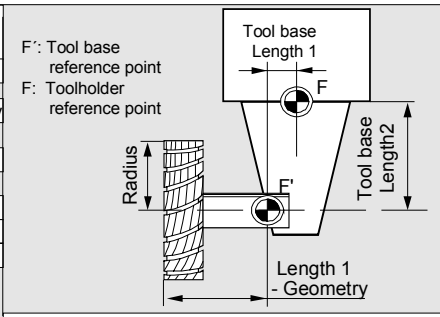
The tool offsets can be entered not only via the operator panel front but also via the data input interface.

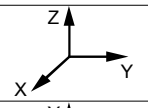
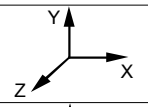
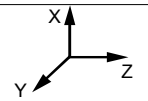
Calculation of tool base dimensions for two-dimensional millhead

Entries in tool parameters	
DP1	5xy
DP3	Length 1 - Geometry
DP6	Radius - Geometry
DP21	Length 1 - Base
DP22	Length 2 - Base
DP23	Length 3 - Base

Wear values according to requirements

Other values must be set to zero



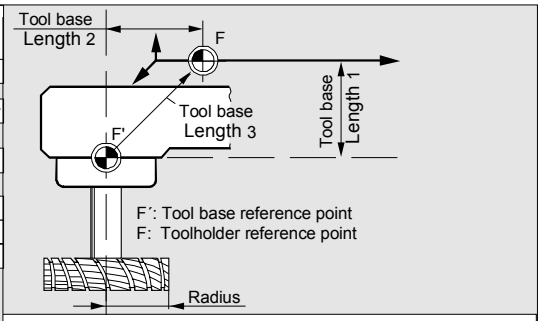
Effect		
G17:	Length 1 in Z Length 2 in Y Length 3 in X Radius in Y/Z	
G18:	Length1 in Y Length 2 in X Length 3 in Z Radius in X/Y	
G19:	Length 1 in X Length 2 in Z Length 3 in Y Radius in Z/X	

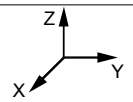
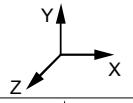
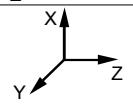
Calculation of tool base dimensions for three-dimensional millhead:

Entries in tool parameters	
DP1	5xy
DP3	Length 1- Geometry
DP6	Radius - Geometry
DP21	Length 1 - Base
DP22	Length 2 - Base
DP23	Length 3 - Base

Wear values according to requirements

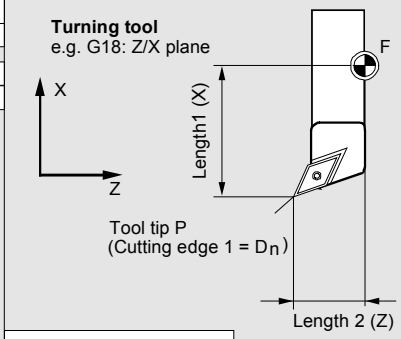
Other values must be set to zero



Effect		
G17:	Length 1 in Z Length 2 in Y Length 3 in X Radius in Y/Z	
G18:	Length 1 in Y Length 2 in X Length 3 in Z Radius in X/Y	
G19:	Length 1 in X Length2 in Z Length 3 in Y Radius in Z/X	

Required length compensation values for turning tools:

Entries in tool parameters	
DP1	5xy
DP3	Length 1
DP4	Length 2



Wear values according to requirements

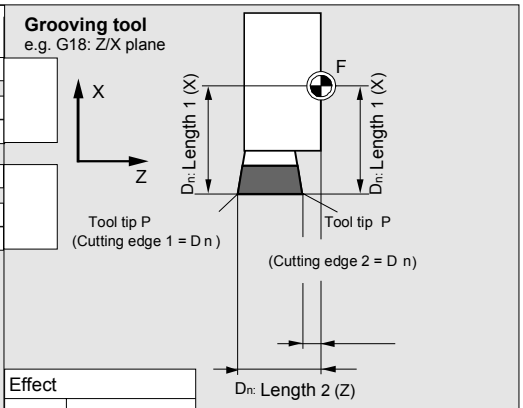
Other values must be set to zero

Effect	
G17:	Length 1 in Y Length 2 in X
G18:	Length 1 in X Length 2 in Z
G19:	Length 1 in Z Length 2 in Y

F: Toolholder reference point

Turning tool with several cutting edges – length compensation:

Entries in tool parameters	
DP1	5xy
DP3	Length 1
DP4	Length 2
DP1	5xy
DP3	Length 1
DP4	Length 2



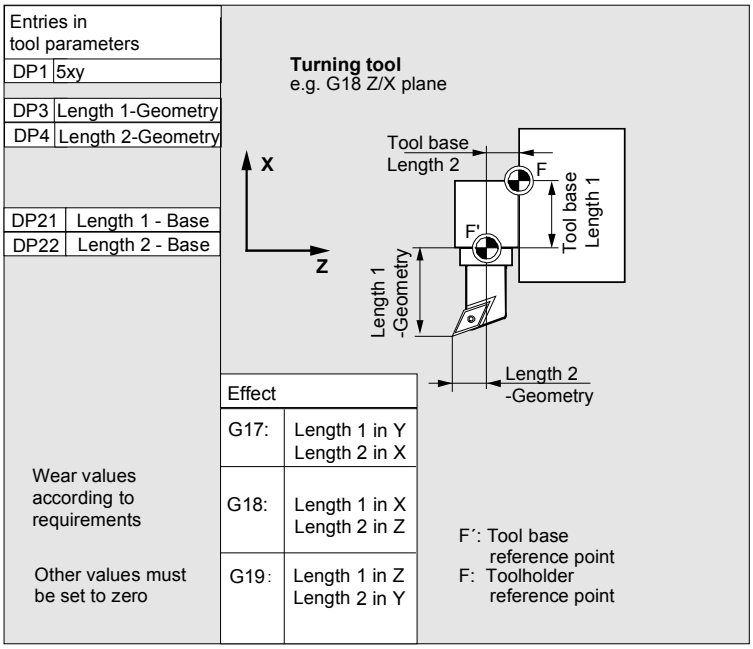
Wear values according to requirement

Other values must be set to zero

Effect	
G17:	Length 1 in Y Length 2 in X
G18:	Length 1 in X Length 2 in Z
G19:	Length 1 in Z Length 2 in Y

F: Toolholder reference point

Calculation of tool base dimensions for turning machine:



Tool type 4xy (grinding tools) is calculated separately.

For the geometric values (e.g. length or radius), there are several entry components.

Parameters	Grinding wheel comp. left	Grinding wheel comp. right	Dresser left	Dresser right
Tool-specific parameters				
$TC_DP1	Tool type	*(2^0=1)	Tool type	Tool type
$TC_DP2	Length of cutting edge	Length of cutting edge	Length of cutting edge	Length of cutting edge
Geometry tool length compensation				
$TC_DP3	Length 1	*(2^2=4)	Length 1	Length 1
$TC_DP4	Length 2	*(2^3=8)	Length 2	Length 2
$TC_DP5	Length 3	*(2^4=16)	Length 3	Length 3
$TC_DP6	Radius	Radius	Radius	Radius
$TC_DP7 to $TC_DP11	Reserved	Reserved	Reserved	Reserved
Wear tool length compensation				
$TC_DP12	Length 1	*(2^{11}=2048)	Length 1	Length 1
$TC_DP13	Length 2	*(2^{12}=4096)	Length 2	Length 2
$TC_DP14	Length 3	*(2^{13}=8192)	Length 3	Length 3
$TC_DP15	Radius	Radius	Radius	Radius
$TC_DP16 to $TC_DP20	Reserved	Reserved	Reserved	Reserved
Base dimension/adapter dimension tool length compensation				
$TC_DP21	Basic length 1	*(2^{20}=1048576)	Basic length 1	Basic length 1
$TC_DP22	Basic length 2	*(2^{21}=2097152)	Basic length 2	Basic length 2

$TC_DP23	Basic length 3	*(2^{22}=4194304)	Basic length 3	Basic length 3

Technology				
$TC_DP24	Reserved	Reserved	Reserved	Reserved
$TC_DP25	Reserved	Reserved	Reserved	Reserved
Additional parameters				
$TC_DPC1				
up to				
$TC_DPC10				

* Value of the chaining parameter if the compensation parameter is to be chained.

Parameter no. for radius calculation $TC_TPG9

With this parameter it is possible to define which offset value is used for grinding wheel surface speed, tool monitoring and centerless grinding. The value always refers to cutting edge D1.

$TC_TPG9 = 3	Length 1 (geometry + wear + base, depending on tool type)
$TC_TPG9 = 4	Length 2 (geometry + wear + base, depending on tool type)
$TC_TPG9 = 5	Length 3 (geometry + wear + base, depending on tool type)
$TC_TPG9 = 6	Radius

*: The tool parameter of cutting edge 2 is chained to the parameter of cutting edge 1 (see tool-specific grinding data $TC_TPG2, chain rule). Here, typical chains are shown and the associated place value is specified in brackets.

Spindle number $TC_TPG1

This parameter contains the number of the spindle to which the monitoring data and GWPS refer.

Chain rule $TC_TPG2

This parameter defines which tool parameters of the right wheel edge (D2) and left wheel edge (D1) must be chained (see TOA data). If the value of one of the chained parameters is changed, it is then automatically included in the chained parameter.

It must be noted that the minimum grinding wheel radius must be specified in the Cartesian coordinate system for an inclined grinding wheel. The length compensations always specify the distances between the tool holder reference point and the tool tip in Cartesian coordinates.

The monitoring data apply to both the left-hand and the right-hand cutting edge of the grinding wheel.
The tool lengths are not automatically compensated when the angle is altered.
On inclined axis machines the same angle must be specified for the inclined axis and the inclined wheel.

Offsets that are not required must be assigned the value 0 (= default when the offset memory is set up).
The tool offsets can be entered not only via the operator panel front but also via the data input interface.

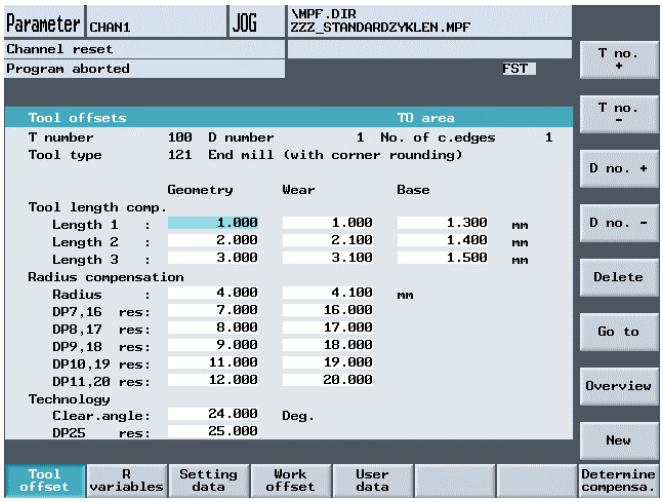
References

For programming of compensation data see
/PG/ Programming Guide Fundamentals

5.2 Tool offset

5.2.1 Tool offset function and basic display

Tool offset data consist of data that describes the geometry, wear, identification, tool type, and the assignment to parameter numbers.
The unit used for the dimensions of the tool is displayed.
The input field is highlighted.
If no tool management, e.g. Standard, ShopMill, ShopTurn is available, the following window appears after selecting the “Parameters” operating area:



Every offset number contains up to 25 parameters depending on the tool type.
The number of parameters shown in the window is that for the tool type.

The maximum number of offset parameters (T and D numbers) can be set by means of machine data.

Machine manufacturer Please note information supplied by machine tool manufacturer!

Horizontal softkeys

You can select different data types with the horizontal softkeys:

Tool offset	Selection of "Tool offset" menu
R variables	Selection of "R variables" menu
Setting data	Selection of "Setting data" menu
Work offset	Selection of "Work offset" menu
User data	Selection of "User data" menu
Determine compensa.	Support in determining tool offsets. This softkey is not required if the tool management function is available.

Vertical softkeys

The vertical softkeys support data input:

T No. +	Selection of the next tool
T No. –	Selection of the previous tool
D No. +	Selection of next higher offset number (cutting edge)
D No. –	Selection of next lower offset number (cutting edge)
Delete...	Deletion of a tool or cutting edge
Go to	Find any tool or the active tool
Overview	List of all available tools
New	New cutting edge or a new tool

5.2.2 Create new tool

Function

If you create a new tool, the relevant tool types are automatically pre-selected as input support when you select the tool group.

Operating sequence

The "Tool offset" window is displayed automatically.

New

New tool

Press "New" and "New tool" softkey.
The "New tool" window appears on the screen.
As soon as you enter the first characters in the string for the tool group, for example

- 5xy Turning tools

all available tool types in the 5xy group are automatically displayed for you to select, i.e.

- 500 Roughing tool
- 510 Finishing tool
- 520 Plunge cutter
- 530 Parting tool
- 540 Threading tool

Enter the digits for your selections via the alphanumeric keypad or select them from the displayed list.

No new tool is set up. Input is discarded.

Abort

OK

The new tool is created.
The window is closed and the tool list is overlaid.

5.2.3 Display tool

Function

You can select tools that you have set up and access their tool compensation data.

Operating sequence

The "Tool offset" window is displayed automatically.

If the "Parameter" area has already been selected, the window and the last tool selected when the area was exited are displayed.
The tool offset data of the current tool are displayed immediately. If no tool has yet been selected, the data of the first tool are shown together with its first D number.
If no tools are available in the area, a message is output.

T No. +	T No. –

Select the set up tools.

Further information

Input of the geometry and wear data of the tool can be disabled using the key switch.

5.2.4 Find tool

Function

There are two methods by which you can find tools and view their tool offset data.

Operating sequence

Find tool by selecting softkey "Overview":
The "Tool offset" window is displayed automatically.

Overview

The "Overview" softkey displays a list of all existing tools.

OK

Position the cursor on the tool you wish to find and confirm your request by pressing the "OK" softkey.

The new tool is selected and displayed in the "Tool offset" window.

Find tool by selecting softkey "Go to":

Go to

Softkey "Go to" opens a window in which you can enter the T or D number you wish to find or in which you can select the pre-selected or active tool using the vertical softkeys.

Preselect T/D No. or Active T/D No.

OK

The tool you are searching for is positioned with "OK". Its tool offsets are then displayed.

5.2.5 Delete tool

Function

The tool is deleted together with all its cutting edges and the tool list updated accordingly.

Operating sequence

The "Tool offset data" window is displayed automatically.

T No. + T No. –

Scroll until you reach the tool to be deleted.

Delete

The vertical softkey bar changes when softkey "Delete" is selected.

Delete tool

Press the "Delete tool" softkey.
The tool and all its edges are deleted. The tool offsets of the tool number preceding the deleted tool are displayed.

5.2.6 Create new cutting edge

Function

To help you to select a new tool edge, the associated tool types are displayed automatically when you select a tool group.

Operating sequence

The "Tool offset" window is displayed automatically.

New

New edge

Press "New ..." and "New edge" softkeys.
The window "New edge" is displayed.
As soon as you enter the first characters in the string for the tool group, for example

- 5xx Turning tools

all available tool types in the 5xx group are automatically displayed for you to select, i.e.,

- 500 Roughing tool
- 510 Finishing tool
- 520 Plunge cutter
- 530 Parting tool
- 540 Threading tool

Abort

Press "Cancel" to discard entered values.

OK

Press "OK" to save the entered values.

5.2.7 Display cutting edge

Function

You can select and modify the edges of tools that you have set up.

The "Tool offsets" window is selected.

T No. +

T No. –

Select a tool of your choice and a cutting edge.

5.2.8 Find cutting edge

Overview

Go to

Function

You can use one of two search methods:

1. Overview: List of existing tools with their edges is displayed.

2. Go to: Opens a window in which you can enter the T or D number you wish to find or you can use softkeys to select the "Pre-selected T/D No." or the "Active T/D No.".

5.2.9 Delete cutting edge

D No. +

D No. -

Delete

Delete tool edge

Function

You can delete one or several edges of a tool. The tool list is updated automatically.

Select the tool edge, press

"Delete" softkey and

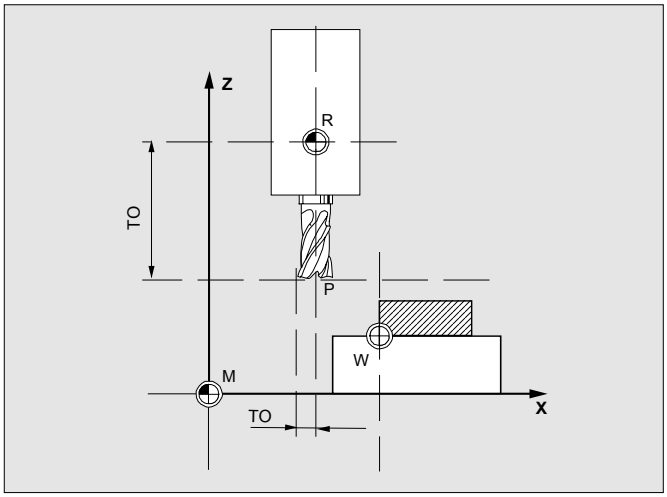
the "Delete tool edge" softkey.

The displayed tool edge is deleted.

5.2.10 Determine tool offsets

Function

The "Determine tool offsets" function allows you to change the reference values of different axes and then to calculate them.



TO Tool offset, absolute dimension
R Tool mounting point
M Machine zero
W Workpiece zero

Operating sequence

The "Tool offset" window is displayed automatically.
Position the cursor on the tool parameter you wish to change.

Determine compensa.

The "Absolute dimension" window opens.

SELECT

Select the appropriate axis with the "Select key". Alter the reference value if necessary using the numeric keypad.

OK

When you press softkey "OK", the current position and corresponding reference value for the selected tool parameter are calculated.
The following equation applies: Position – reference value = input value
The window is closed.

Calculate

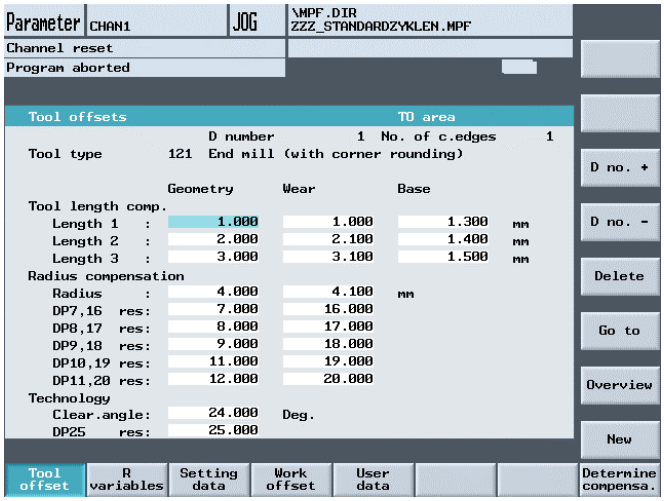
Position – reference value is entered in the input field. The window remains open.

If "Jog" mode is selected, it is also possible to change the position by traversing the axes.
The control automatically calculates the value from the reference value and the new position.

5.2.11 Tool offsets with D-numbers only (flat D no.)

Function

It is possible to specify that tools with a flat D number can only be selected via their flat D number. One D number can be assigned only once per tool, i.e. each D number represents exactly one offset data record.



Machine manufacturer Please note information supplied by machine tool manufacturer!

References /FB1/W1: Tool offset

5.2.12 Make active tool offset operative immediately

Function

The machine data can be set to specify that the active tool offset can be activated immediately if the part program switches to the "Reset" or "Stop" state.

Further information

When the function is used in the Reset state, the machine data must be set such that the offset is not reset when the program switches to Reset.

Machine manufacturer Please note information supplied by machine tool manufacturer!

References /FB1/K2: Axes/Coordinate Systems etc.

Caution

The offset is applied in the next programmed axis motion in the part program after "Cycle Start" in Reset.

5.3 Tool management

The tool management system is organized by means of various configurable lists which show different views of the tools used.

Magazine list

In the "Magazine list", the tools of a magazine are displayed in order of ascending magazine location numbers.

You can find, display and, in the majority of cases, modify the data.

This list is mainly used to load and unload tools during setup, and to move tools between magazines.

Tool list

In the "Tool list", the tools are displayed in the order of ascending T numbers.

You use this list if you are working with small tool magazines and know the exact location of each tool in each of the magazines.

ShopMill tool management

You can use the ShopMill tool management as an alternative to the standard tool management. It uses workshop-compatible tool management for **milling machines**. You can select it via an MD.

ShopTurn tool management

You can use the ShopTurn tool management as an alternative to the standard tool management. It uses workshop-compatible tool management for **turning machines**. You can select it via an MD.

Machine manufacturer

For details of the functionality of your tool management system, please refer to the machine manufacturer's instruction manual.

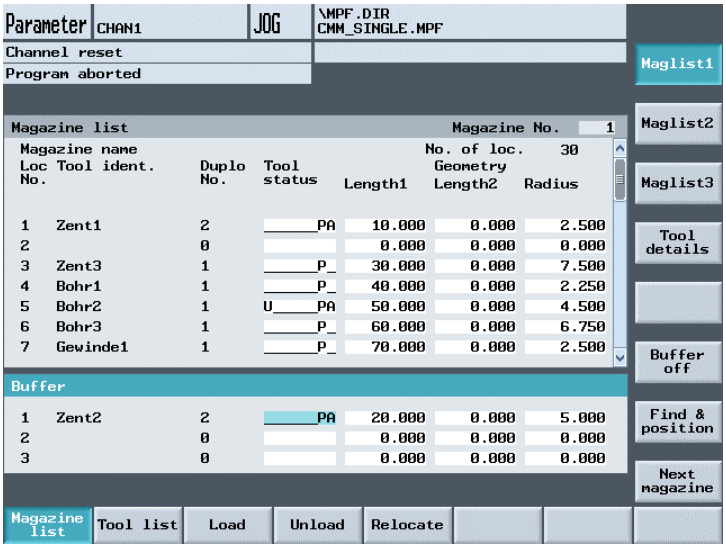
References

/IAM/IM2/Installation & Start-Up Guide HMI Embedded
/FBW/Description of Functions Tool Management or
/FBSP/Description of Functions ShopMill
/FBT/Description of Functions ShopTurn

5.3.1 Tool management basic display

You can manage your tool magazines in the "Magazine list" basic display.

The tool management offers the most commonly used tools as tool types. You can assign geometric and technological data to the tool types in order to set up your master tool data. There can still be several versions of each tool. You can assign the actual data of the tool used (particular tool data) to these versions.



The tool management basic display contains the current "magazine list" with the following information:

Loc. No. Location number

Tool designation Name of tool
Other displays configured by the machine manufacturer, e.g.

Duplo no. Number of spare tool (replacement tool)

Tool status

Tool status
No display = Replacement tool
A = Active tool
F = Tool enabled
G = Tool disabled
M = Measured tool
V = Warning limit reached
W = Tool is being changed
P = Fixed-location-coded tool
E = Tool was in use

WZ type

Tool type
Depending on the tool type, only certain tool offsets are enabled for selection when the tool is loaded. All other tool types are preset with the value "0".

T number

Internal T numbers which may be needed for reloading tool data.

Geo - L1 ... Radius …

Tool offsets such as length, radius, wear, monitoring data, etc.

Horizontal softkeys

Magazine list

The "Magazine list" basic display shows all the tools that are already assigned to a magazine location ("loaded").

Tool list

All tools which are stored as a set of data on the NC are displayed (irrespective of whether or not they have been assigned to a magazine location).

Load

A magazine location is assigned to the tool.

Unload

The tool is deleted from the current magazine location.

Relocate

The tool is moved from the current magazine location to another location.

Vertical softkeys

Selection of user-specific windows (if configured), e.g.

(Names assigned by user)

Maglist 1
Maglist 2
Maglist 3

- General data
- Geometry tool data
- Wear data

Tool details

Display and edit the tool offset data for a tool.

Buffer on or Buffer off

Display and skip the buffer window in the magazine list. The display shows spindles, grippers, etc., i.e. locations which can accommodate tools but which are not magazine locations.

Find & position

You can use this softkey to find a tool or tool location in the active tool magazine and position the tool. The tool is moved to the loading point.

Next magazine

Advance to the next magazine.

5.3.2 Select new tool

Function

If you want to load or unload a tool to/from a magazine, alter or re-edit the current tool data or create a new tool edge, you must first select the appropriate tool in the "Magazine list" or "Tool list".

Operating sequence

Tool management

Select softkey "Tool management".
The horizontal and vertical softkey bars change.

Select the menu via the softkey

"Magazine list" or "Tool list".

Magazine list or Tool list

Next magazine

Select the appropriate magazine.

Position the cursor bar on the appropriate tool.
The tool is now selected for editing.

5.3.3 Display and modify tool data

Function

You can view and edit the tool data of the tool selected in the "Magazine list" or "Tool list".

You can edit the following tool edge data:

- offset values
- Monitoring data

Operating sequence

Magazine list or Tool list

"Magazine list" or "Tool list".

Position the cursor bar on the appropriate tool.
The appropriate tool is selected.

Tool details

Select the "Tool details" softkey.
The "Tool data" menu is displayed.
The vertical softkey bar changes again.

Display/edit tool data:
You can display and edit tool data in menus:

- "Magazine list" (if the tool data, edge data and tool user data are configured in the magazine list)

D No. +

Switches forward by one edge.

D No. -

Switches back by one edge.

New edge

Creates new edge.
Edit compensation data of new edge.

Abort

Abort input.

OK

Confirm and accept input.

A new cutting edge can be attached to a tool at any time (even if the corresponding tool is already in the magazine). Here you enter the cutting edge data.

5.3.4 Load tool

Function

A tool is loaded to a magazine at the magazine location on which the cursor bar is positioned in the "Magazine list" or "Tool list" menu.
You can load a tool to a magazine in one of the following ways:

- **Loading from the "Magazine list"**
  You can load all tools in the magazine. The associated tool data can be loaded from the master data catalog or code carrier (if available) or entered manually as required.

- **Load from "Tool list"**
  You can load magazines whose data are already stored in the TO memory (if available).

Operating sequence

The "Magazine list" menu is displayed.
The horizontal and vertical softkey bars change.

Load from "Magazine list":

Magazine list

The "Magazine list" menu is selected.
The appropriate magazine is selected.

Load

Press the "Load" softkey.
The vertical softkey bar changes.

There are three ways to search the empty location according to different tool sizes in conjunction with location types:

1. Find an empty location
2. Current location
3. Load location

1. Enter the "Tool size" and "Location type" in the dialog box.

Find empty location

If more than one loading point is configured, select the desired loading point from a query window.
The system searches for a suitable empty location.
The cursor bar is automatically positioned on the magazine location found in the "Magazine list".

2. You have found an empty location in front of the current loading point.

Current location

When you select "current location," the tool is loaded to the empty location in front of the loading point.
The cursor bar is automatically positioned in the "Magazine list" on the magazine location found in front of the loading point.

3. Position the cursor on the location of your choice in the magazine list.

Load location

"Load directly to spindle" is possible if the cursor is positioned on the spindle magazine location.

Use the input key to accept the entries for the tool identifier, duplo No. etc.

Abort

Abort the loading operation.

OK

Confirm your inputs with "OK".

Load from "**Tool list**":

Tool list

The "Tool list" menu is selected.
The appropriate tool is selected.

Load

Press the "Load" softkey.
The vertical softkey bar changes.

Find empty location

After pressing the "Find empty location" softkey, the system searches for a suitable location and displays it.

Abort

Abort the loading operation.

OK

Initiate the loading operation.

The location found is entered under the location number.
If data are still missing, the tool data display is opened with the missing data set to their defaults. You can start the loading operation again.

5.3.5 Unloading of a tool

Function

This function allows you to unload a selected tool and to save its data.

Operating sequence

Unload from "Magazine list":

Tool list

The "Magazine list" menu is selected.
The appropriate magazine is selected.
The appropriate tool is selected.

Unload

When you press the "Unload" softkey, the magazine number and load point are displayed in the "Unload" window.

If more than one load point is configured, select the appropriate load point from the selection window.

OK

When you press "OK," the tool is unloaded.

Abort

Press "Abort" to stop the "Unload" process.

"Unload directly from spindle" is possible only if the buffer is selected and the cursor is positioned on the location of the spindle.

Unload from "Tool list":

Tool list

The "Tool list" menu is selected.
The appropriate tool is selected.

Unload

When you press the "Unload" softkey, the "Unload" window opens and the magazine number and load point are displayed.

If more than one loading point is configured, select the appropriate loading point from the selection window.

OK

Click "OK" to begin the unloading process or

Abort

Click on "Abort" to abort the process.

Delete tool

The tool data of the selected tool is deleted from the TO memory. You can do this only if the tool is not currently stored in a magazine location.

5.3.6 Relocating a tool

Function

This function allows you to move a selected tool from one location to another.

Operating sequence

Magazine list

The "Magazine list" menu is displayed.

Relocate

Position the cursor on the tool to be relocated in the magazine list. When you select the "Relocate" key, the "Relocate tool" window opens.

There are 2 methods by which you can select the new empty location for the tool:

1. Enter the magazine and location numbers in the "Relocate tool" window.
2. Select softkey "Find empty location" and an appropriate empty location is suggested.

Or Find emtpy location

OK

Press "OK" to save the tool to the new empty location,

Abort

and the relocation operation is aborted with "Abort".

Use magazine number 9998 to move a tool to or from the spindle.

5.3.7 Finding and positioning tools

Function

You can find a tool or tool location in the active tool magazine and position the tool.

Magazine list

The "Magazine list" menu is displayed.

Operating sequence

Next magazine or Maglist x

Select the required magazine.

Find/ position	Press the "Find/position" softkey.
	A dialog box is overlaid. Enter the tool identifier e.g. T501, duplo No. e.g. 2, as well as the load point.
OK	Press "OK" to execute. A message appears: e.g. "location 1 found".
Find/ position	Continue to press the "Find/position" softkey.
	The vertical softkey bar changes.
Find location	Press the "Find location" softkey. The system suggests the location.
Position	Press the "Position" softkey. The tool/location is moved to the load point. If there are several load points, a window opens in which you can select the appropriate point with the cursor.

5.4 ShopMill tool management

Function

ShopMill tool management allows workshop-compatible tool management of milling machines.
The following lists are available to you for this function:

- Tool list
- Tool wear list
- Magazine list

You enter the tools and their offset data in the tool list/tool wear list.
You can see from the magazine list which magazine locations are disabled or not.

Tool list

The tool list displays all tools and their offset data stored as a tool data block in the NC, irrespective of whether they are assigned to a magazine location. The tool list offers the current tool types for which geometric and technological data can be assigned.

Load/Unload
When a tool is loaded, it is taken to a magazine location.
Unloading removes the tool from the magazine.

Sorting
The tools in the tool list and tool wear list can be sorted according to magazine location, name, and type.

Machine manufacturer

Machine data can be set to hide the "Load," "Unload" and "Sort" softkeys.

Manual tools
Manual tools are included in the tool list, but not stored in the magazine. They must be attached to the spindle by hand.

Tool wear list

This list specifies which wear data (length and radius/diameter) are to be taken into account. The following types of monitoring can also be defined for a tool:

- Monitoring of the effective operating time (tool life)
- Monitoring of number of tool load operations
- Additional tool status data (disable tool, tool in fixed location, oversized tool)

Fixed/flexible assignment of locations
You can define via a machine data whether all tools are fixed or variable location coded.

- With fixed location coding, the tool is permanently assigned to a magazine location. This concept can be used for machines with disk-type magazine.
- With variable location coding, a tool can also be conveyed to a magazine location other than the original location. This concept can be used for machines with chain magazine. Individual tools can be set to fixed-location-coded in the tool wear screen display at the operator interface.

Magazine

The magazine locations are listed with their tools, magazine locations are indicated as disabled/not disabled, and the properties assigned to the active tool (e.g. oversize) are displayed in the magazine list.

References

/FBW/ Description of Functions Tool Management or
/FBS/ Description of Functions ShopMill

/BAS/ Operation/Programming ShopMill

5.4.1 Function scope

Function

ShopMill tool management supports the following tool types, tool parameters and magazine parameters:

Tool types

– 120 End mill
– 200 Twist drill
– 220 Center drill
– 710 3D probe
– 711 Edge probe
– 110 Cylindrical die sinker
– 111 Ballhead cutter
– 121 End mill with corner rounding
– 155 Bevel cutter
– 156 Bevel cutter with fillet
– 157 Conical die sinker

Tool parameters

– Magazine location/magazine number
– Tool type
– Tool name
– Duplo number
– Geometry length 1
– Geometry radius
– Wear length 1
– Wear radius
– Type of tool monitoring: Tool life
Number of workpieces
– Tool status: Tool disabled
– Tool status: Tool oversized (right and left half locations)
– Tool status: Tool in fixed location
– Fillet radius
– Angle for conical milling tools

Magazine parameters

– Magazine location disabled

5.4.2 Select tool list

Tool list

Operating sequence

If "ShopMill" tool management is set up, the tool list menu is automatically displayed when you call up the "Parameters" operating area. Otherwise you can call the tool list via softkey.

5.4.3 Create new tool

Function

You create new tools in the tool list. A selection of tool types is displayed for this purpose. The tool type determines which geometry data are required and how they will be computed. The following common tool types are available:

Operating sequence

Tool list

Select the "Tool list" softkey.

New tool

Press the "New tool" softkey.

The vertical bar changes and you can select different tools via the softkey.

▲ ▼

Select the tool location of your choice with the cursor keys

and select the softkey for the desired tool type.

Cutter ... 3D tools

More

Additional tool types are available via the "More" softkey.

The new tool is created.

3D tools

In the case of 3D tools, you must define parameters in addition to the geometry data in the tool list.

Type	Name	Additional parameters
110	Cylindrical die mill	-
111	Ball end mill	Smoothing radius
121	End mill with corner rounding	Smoothing radius
155	Truncated cone mill	Angle for conical tools
156	Bevel cutter with corner rounding	Rounding radius, angle of conic. tools
157	Tapered die mill	Angle for conical tools

Details

Press the softkey "Details" and enter the fillet radius and angle for the conical milling tools.

5.4.4 Set up more than one edge for each tool

In the case of tools with more than one cutting edge, a separate set of offset data is assigned to each cutting edge. You can create up to 9 cutting edges for each tool.

In the case of ISO programs (e.g. ISO dialect 1) you must specify an H number. This corresponds to a particular tool offset set.

Operating sequence

Follow the instructions given above to set up tools with more than one edge in the tool list and enter the offset data for the 1st edge.

Edges ... New egde

Then select the "Edges" and "New edge" softkeys.

Instead of the input fields for the first cutting edge, the offset data input fields for the second cutting edge are displayed.

Enter the offset data for the second cutting edge.

Repeat this process if you wish to create more tool edge offset data.

Delete edge

Select the "Delete edge" softkey if you want to delete the tool edge offset data for an edge.
You can only delete the data for the edge with the highest edge number.

D No + ... D No -

By selecting softkey "D No. +" or "D No. –", you can display the offset data for the edge with the next higher or next lower edge number respectively.

5.4.5 Changing a tool name

A tool that has just been created in the tool list is automatically assigned the name of the selected tool group. You can change this name as often as you want to

- A tool name, e.g. "plain mill_120mm" or
- A tool number, e.g. "1".

The tool name must not exceed 17 characters in length. You can use letters, digits, the underscore symbol (_), periods (".") and slashes ("/").

5.4.6 Creating duplo/replacement tools

The duplo/replacement tool is a tool that can be used for the same machining operations as a tool that already exists (e.g. for use after a tool breakage).

When you create a replacement tool, you must use the same name as is used for a comparable tool.

Operating sequence

Create the replacement tool as a new tool. See Subsection "Create new tool"

Assign the same name as the original tool to the replacement tool. Confirm the name with the "input" key and the duplo number of the replacement tool is automatically incremented by 1.

The sequence for inserting a replacement tool is determined by the duplo number **DP**.

5.4.7 Manual tools

The "Manual tool" function must be set up by the machine manufacturer.

Manual tools are tools which are required during machining, but are only available in the tool list but not in the tool-holding magazine. These tools must be attached/detached manually to/from the spindle.

Machine manufacturer Please note information supplied by machine tool manufacturer!

5.4.8 Enter tool wear data

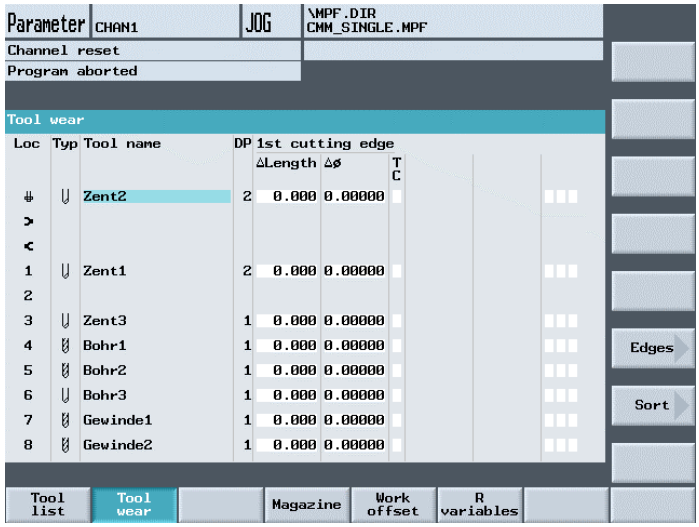
Tools that are in use for long periods are subject to wear. You can measure this wear and enter it in the tool wear list. ShopMill then takes this information into account when calculating the tool length or radius compensation. This ensures a consistent accuracy in workpiece machining.
You enter the wear data for a tool that you have created in the tool wear list.

The upper limits are set in a machine data code.

Operating sequence

Tool wear

Select the "Tool wear" softkey.



Example of a tool wear list with variable location allocation

Place the cursor on the tool whose wear data you want to enter.

Enter the differences for length (ΔLength X, ΔLength Z) and radius/diameter (ΔRadius/Δ∅) in the appropriate columns.

The wear data entered is added to the radius but subtracted from the tool length. A positive differential value for the radius therefore corresponds to an oversize (e.g. for subsequent grinding).

Machine manufacturer

Please note information supplied by machine tool manufacturer!

5.4.9 Activate tool monitoring

ShopMill allows you to monitor the tool life of the tools automatically to ensure constant machining quality.

You can also disable tools that you no longer want to use or identify them as oversize or assign them to a magazine location.
In the tool wear list you can assign each tool the following tool monitoring and properties:

- Tool life
- Number of workpieces
- Other tool properties
  - Lock tool
  - Tool in fixed location
  - Oversized tool

The tool monitoring functions are activated via machine data.

Machine manufacturer Please follow the machine manufacturer's instructions!

Operating sequence

Select "Tool wear" via softkey

Tool wear

Tool life (T) The tool life is used to monitor the service life of a tool with machining feedrate in minutes. When the remaining tool life is ≤ 0, the tool is set to "disabled". The tool is not put into operation on the next tool change. If a replacement tool is available, it is inserted in its place. The tool life monitoring function always refers to the selected tool edge.

Count (C) With the count C, however, the number of tool changes is counted in the spindle. The tool is also disabled in this case, when the remainder reaches "0".

Wear (W) With wear W, the greatest value in the wear parameters Δ Length X, Δ Length Z or Δ Radius or Δ Δ in the wear list is monitored. Here too the tool is disabled if one of the wear parameters reaches the value for wear W.

Machine manufacturer Please follow the machine manufacturer's instructions!

Prewarning limit	The pre-warning limit specifies the tool life, workpiece count or wear at which the first warning is displayed.
Disabled (G)	Individual tools can also be disabled manually if you no longer want to use them for workpiece machining.
Oversize (U)	In the case of oversize tools, neighboring magazine locations (left and right adjacent location) are only reserved alternately, i.e. you can only insert the next tool in the next magazine location but one (this can also contain an oversize tool).
Coded for fixed location (P)	You can assign tools a fixed location, i.e., the tool can only be used at the current magazine location. The tool returns to the old magazine location when switching back.

Monitoring tool use

Tool wear

Select the "Tool wear" softkey.

Position the cursor on the tool that you want to monitor.

In the column "T/C" select the parameter that you wish to monitor (T = Tool life, C = Count, W = Wear).

Enter a pre-warning limit for tool life, count or wear.

Enter the scheduled service life for the tool, the scheduled number of workpieces to be machined or the maximum permissible wear.

The tool is disabled when the tool life, count or wear is reached.

Enter tool status

Tool wear

Select the "Tool wear" softkey

Place the cursor on a tool.

Option G	Select the option "G" in the first field of the last column if you want to disable the tool for machining.
	-or-
Option U	Select the option "U" in the second field of the last column if you want to mark the tool as oversize.

-or-

Option P

Select the option "P" in the third field of the last column if you want to assign the tool to a fixed magazine location.

The set tool properties are immediately active.

5.4.10 Managing magazine locations

The magazine locations are listed with their tools, magazine locations are indicated as disabled/not disabled, and the properties assigned to the active tool (e.g. oversize) are displayed in the magazine list.

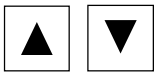
Operating sequence

Magazine

Select the "Magazine" softkey.

Disabling a magazine location

Magazine locations can be reserved or disabled for specific tools, e.g. in the case of an oversized tool.



Selecting the required magazine location.

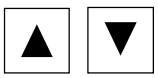
Toggle in column "Disable location" with the softkey "Alternative" until a "G" (= disabled) appears in the field in question. The location is now disabled. A tool can no longer be loaded into this magazine location.

Tool status

In the "Tool status" column you can see which properties have been assigned to the active tool:

- G: Tool is disabled
- U: Tool oversized
- P: Tool at a fixed location

Enabling a magazine location



Position the cursor on the disabled field.

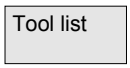
Deselect option G in the "Location disable" column.
The magazine location is enabled again.

5.4.11 Delete tools

Function

Tools can be deleted from the tool list.

Operating sequence



Select the "Tool list" softkey.

Select the tool of your choice.

Press the "Delete tool" softkey and

confirm with "Delete".

The tool data of the selected tool are deleted, and the magazine location in which the deleted tool was located is enabled.

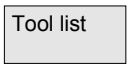
The tool is not deleted with "Abort".

5.4.12 Change tool type

Function

In the tool list you can change a tool type into another tool type.

Operating sequence



Select the "Tool list" softkey.

Select the desired tool and position the cursor on input field "Type".

You can switch to the tool type you want with the "Alternative" key. The input fields for the new tool type are displayed.

5.4.13 Load or unload tool in magazine

Loading and unloading of tools into and out of magazine locations must be enabled in a machine data code.

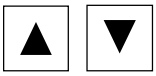
Function

You can unload tools in the magazine that you are not using at present. ShopMill then automatically saves the tool data in the tool list outside the magazine. If you want to use the tool again later, simply load the tool with the tool data into the corresponding magazine location again. Then the same tool data does not have to be entered more than once.

Machine manufacturer Please follow the machine manufacturer's instructions!

Operating sequence

Load a tool into the magazine



Place the cursor on the tool that you want to load into the magazine (if the tools are sorted according to magazine location number you will find it at the end of the tool list).

Press the "Load" softkey.

The "Empty location" window appears. The "Location" field is initialized with the number of the first empty magazine location.

Press the "OK" softkey to load the tool into the suggested location.

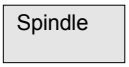
-or-

Enter the location number you require and press the "OK" softkey.

-or-

Enter the location number you require and press the "OK" softkey.

-or-



Press the "Spindle" and "OK" softkeys to load a tool into the spindle.

The tool is loaded into the specified magazine location.

Unload an individual tool from the magazine

Select the "Magazine" softkey.

Position the cursor on the tool that you want to unload.

Press the "Unload" softkey.

The tool is unloaded from the magazine.

Unload all tools from the magazine

Select the "Magazine" softkey

Press the "Unload all" and "Unload" softkeys.

All tools are unloaded from the magazine.

You can abort the unloading process at any time by pressing the "Abort" softkey. The current tool is still unloaded, but then the process is aborted.
The unloading process is also aborted if you exit the magazine list.

5.4.14 Relocating tools

Tools can be relocated within magazines or between different magazines, which means that you do not have to unload tools from the magazine in order to load them into a different location.

ShopMill automatically suggests an empty location to which you can relocate the tool. The magazine in which ShopMill searches for an empty location first is stored in a machine data code.
You can also specify an empty magazine location directly or define the magazine ShopMill should search for an empty location.

If your machine has only one magazine, you simply need to enter the location number you require, not the magazine number.

If a spindle location is shown in the tool list, you can also load or unload a tool directly into or out of the spindle.

Machine manufacturer Please follow the machine manufacturer's instructions!

Specify an empty location

Tool list

Select the "Tool list" softkey.

Place the cursor on the tool that you wish to relocate to a different magazine location.

Relocate

Press the "Relocate" softkey.

The "Empty location" window appears. The "Location" field is initialized with the number of the first empty magazine location.

Press the "OK" softkey to relocate the tool to the suggested location.

-or-

Enter the location number you require and press the "OK" softkey.

-or-

Spindle

Press the "Spindle" and "OK" softkeys to load a tool into the spindle.

The tool is relocated to the specified magazine location.

Find an empty location

Relocate

Press the "Relocate" softkey.

The "Empty location" window appears. The "Location" field is initialized with the number of the first empty magazine location.

Enter the magazine number and a "0" for the location number if you wish to search for an empty location in a particular magazine.

-or-

Enter a "0" for the magazine number and location number if you wish to search for an empty location in all magazines.

Press the "OK" softkey.

An empty location is suggested.

Press the "OK" softkey.

The tool is relocated to the suggested magazine location.

5.4.15 Sorting tools in the tool list

Function

The tools can be sorted according to magazine location, tool name (alphabetical), or tool type in the tool list. When you sort according to magazine assignment, the empty locations in the magazine are also displayed.

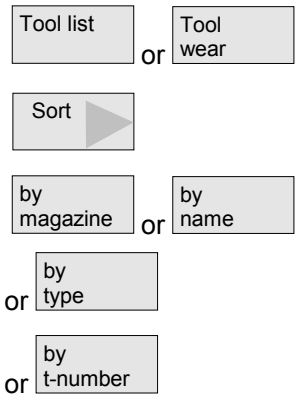
Operating sequence

Select the "Tool list" or "Tool wear" softkey.

Press the "Sort" softkey.

Activate one of the softkeys to choose the sort criteria.

The tools are listed in the new order.



5.5 Tool management ShopTurn

Function

ShopTurn is an operating and programming software program for turning machines that makes it easy for you to operate the machine and to program workpieces.

Various tools are used for machining workpieces. The geometry and technological data of these tools must be known to ShopTurn before you execute your program.
ShopTurn provides the "Tool list", "Tool wear list" and "Magazine list" screen forms for managing your tools,

Tool list

You must enter all the tools that you want to use on the turning machine in the tools list. The tools that are in the tool turret must be assigned to specific magazine locations. You can also sort and delete tools.

Load/Unload
When a tool is loaded, it is taken to a magazine location.
Unloading removes the tool from the magazine.

Sorting
Tools can be sorted in the tool and tool wear lists according to magazine location, name and type.

Machine manufacturer

Machine data can be set to hide "Load", "Unload" and "Sort" softkeys.

Manual tools
Manual tools are included in the tool list, but not stored in the magazine. They must be attached to the spindle by hand.

Tool wear list

You must enter the wear data for your tools in the tool wear list. ShopTurn takes this data into account on machining the workpiece. You can also activate tool monitoring here as well as disable tools or identify them as oversized.

Magazine list

The magazine locations are listed with their tools; magazine locations are indicated as disabled (G) / not disabled ( ), and the properties assigned to the active tool (e.g. oversize (U) are displayed in the magazine list.

References /BAT/ Operator's Guide ShopTurn/Tool Management
/FBW/ Description of Functions Tool Management or
/FBT/ Description of Functions ShopTurn

5.5.1 Function Scope

Function

ShopTurn tool management supports the following tool types, tool parameters and magazine parameters:

You can create up to 250 tools under PCU 20.
You can create up to 98 replacement tools for each tool.

Tool types

- Roughing tool
- Finishing tool
- Recessing tool
- Milling tool
- Drill
- Button
- Limit stop
- Threading tool
- Rotary drill
- 3D_probe

Tool parameters

- Magazine location/magazine number
- Tool type
- Tool name
- Duplo number
- Tool length compensation in the X direction
- Tool length compensation in the Z direction
- Wear radius
- Tip length of a cutting tool or a grooving cutter
- Tip width of a grooving tool
- Angle of teeth for a milling cutter
- Angle of tool tip on a drill
- Type of tool monitoring: optionally according to tool life or tool change in relation cutting edge.
- Tool state: Tool disabled
- Tool state: Tool oversized (right and left half locations)

Magazine parameters

- Magazine location disabled

Additional functionality

- You can use circular magazines that are hidden via display machine data.
- Loading station for loading and unloading tools via display machine data
- Display tools (milling tool/drill) in diameter or radius via display machine data

5.5.2 Select tool list

Operating sequence

Tool list

When you call up the "Parameters" operating area for the first time, the "Tool list" menu is automatically displayed. Otherwise you can call it via softkey.

5.3 Create new tool

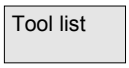
When you want to create a new tool, ShopTurn offers a range of generally available tool types. The tool type determines what geometry data you have to enter and how it is calculated.

Possible tool types

The rotary drill can be used for centric drilling and turning.

Operating sequence

Install the new tool in the tool turret.



Select the "Tool list" softkey.

In the tool list, position the cursor on the location that the tool is to occupy in the turret.

The location must still be empty in the tool list.

Press the "New tool" softkey.

Use the softkeys to select the tool type of your choice.
Additional tool types are available via the "More" softkey.

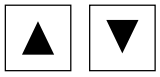
Select the required cutting edge position.

Additional cutting edge positions are available via the "More" softkey.

The new tool is created and automatically assumes the name of the selected tool type.

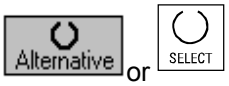
Enter a unique tool name.
You can edit the tool name as required. A tool name may contain a maximum of 17 characters. You can use letters, digits, the underscore symbol (_), periods (".") and slashes ("/").

Enter the offset data of the tool.



If you would like to change the tool's cutting edge position later,

Place the cursor in the "Type" column.



Use the "Alternative" softkey or the "Select" key to select one of the specified options.

5.5.4 Setting up more than one cutting edge for each tool

In the case of tools with more than one cutting edge, a separate set of offset data is assigned to each cutting edge. You can set up a total of 9 edges for each tool.

Operating sequence

Follow the instructions given above to set up tools with more than one edge in the tool list and enter the offset data for the 1st edge.

Edges ... New edge

- Then select the "Edges" and "New edge" softkeys.

Instead of the input fields for the first cutting edge, the offset data input fields for the second cutting edge are displayed.

- Select another cutting edge position if appropriate.
- Enter the offset data for the second cutting edge.
- Repeat this process if you wish to create more tool edge offset data.

Delete edge

- Select the "Delete edge" softkey if you want to delete the tool edge offset data for an edge.
  You can only delete the data for the edge with the highest edge number.

D No + ... D No -

By selecting softkey "D No. +" or "D No. –", you can display the offset data for the edge with the next higher or next lower edge number respectively.

5.5.5 Creating duplo/replacement tools

A replacement tool is one that can be employed to perform the same machining operation as a tool that has already been entered. You can use it, for example, to replace a broken tool.

For each tool in the tool list, you can create several replacement tools. The duplo number 1 is always assigned to the original tool and duplo numbers 2, 3, etc. are assigned to the replacement tools.

Operating sequence

New tool

Create the replacement tool as a new tool. See Section "Create new tool"

Assign the same name as the original tool to the replacement tool.

INPUT

Confirm the name with the "input" key and the duplo number of the replacement tool is automatically incremented by 1.

The sequence for inserting a replacement tool is determined by the duplo number **DP**.

5.5.6 Sort tools

When you are working with large magazines or several magazines, it is useful to display the tools sorted according to different criteria. Then you will be able to find a specific tool more easily in the lists.

Operating sequence

Tool list or Tool wear

Select the "Tool list" or "Tool wear" softkey.

Sort

Press the "Sort" softkey.

by magazine or by name

or by type

Activate one of the softkeys to choose the sort criteria.

The tools are listed in the new order.

5.5.7 Delete tools

Tools that are no longer in use can be deleted from the tool list for a clearer overview.

Operating sequence

Tool list

Select the "Tool list" softkey.

▲ ▼

Select the tool of your choice.

Delete tool

Press the "Delete tool" softkey and

confirm with "Delete".

Delete

The tool data of the selected tool are deleted, the magazine location in which the deleted tool was located is enabled.

Abort

The tool is not deleted with "Abort".

5.8 Load or unload tool in magazine

The tool list has more locations than magazine locations. This means you can unload tools you currently do not need in the magazine and save the tool data in the tool list outside the magazine. If you want to use the tool again at a later point in time, simply load the tool data back to the magazine location. Then the same tool data does not have to be entered more than once.

Loading and unloading of tool data into and out of magazine locations must be enabled in a machine data code.

Machine manufacturer

Please follow the machine manufacturer's instructions!

Operating sequence

Load a tool into the magazine

▲ ▼

Place the cursor on the tool that you want to load into the magazine (if the tools are sorted according to magazine location number you will find it at the end of the tool list).

Load

Press the "Load" softkey.

The "Empty location" window appears. The "Location" field is initialized with the number of the first empty magazine location.

Press the "OK" softkey to load the tool into the suggested location.

-or-

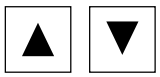
Enter the desired location number and press the "OK" softkey.

The data relating to your tool are now displayed in the specified magazine location.

Unload an individual tool from the magazine

Magazine

Select the "Magazine" softkey



Position the cursor on the tool that you want to unload.

Unload

Press the "Unload" softkey.

The tool data are removed from the magazine and stored in the tool list in a position without a number.

Unload all tools from the magazine

Magazine

Select the "Magazine" softkey

Unload all Unload

Press the "Unload all" and "Unload" softkeys.

All tools are unloaded from the magazine.

You can abort the unloading process at any time by pressing the "Abort" softkey. The current tool is still unloaded, but then the process is aborted.
The unloading process is also aborted if you exit the magazine list.

5.5.9 Relocating tools

Tools can be relocated within magazines or between different magazines, which means that you do not have to unload tools from the magazine in order to load them into a different location.
ShopTurn automatically suggests an empty location to which you can relocate the tool. The magazine in which ShopTurn searches for an empty location first is stored in a machine data code.
You can also specify an empty magazine location directly or define the magazine ShopTurn should search for an empty location.
If your machine has only one magazine, you simply need to enter the location number you require, not the magazine number.

If a spindle location is shown in the tool list, you can also load or unload a tool directly into or out of the spindle.

Machine manufacturer

Please follow the machine manufacturer's instructions!

Specify an empty location

Tool list

Select the "Tool list" softkey.

Place the cursor on the tool that you wish to relocate to a different magazine location.

Relocate

Press the "Relocate" softkey.

The "Empty location" window appears. The "Location" field is initialized with the number of the first empty magazine location.

Press the "OK" softkey to relocate the tool to the suggested location.

-or-

Enter the desired location number and press the "OK" softkey.

-or-

Spindle

Press the "Spindle" and "OK" softkeys to load a tool into the spindle.

The tool is relocated to the specified magazine location.

Find an empty location

Relocate

Press the "Relocate" softkey.

The "Empty location" window appears. The "Location" field is initialized with the number of the first empty magazine location.

Enter the magazine number and a "0" for the location number if you wish to search for an empty location in a particular magazine.

-or-

Enter a "0" for the magazine number and location number if you wish to search for an empty location in all magazines.

Press the "OK" softkey.

An empty location is suggested.

Press the "OK" softkey.

The tool is relocated to the suggested magazine location.

5.5.10 Enter tool wear data

Tools that are in use for long periods are subject to wear. You can measure this wear and enter it in the tool wear list. ShopTurn then takes this information into account when calculating the tool length or radius compensation. This ensures a consistent accuracy in workpiece machining.

Operating sequence

Tool wear

Select "Tool wear" via softkey

Place the cursor on the tool whose wear data you want to enter.

Enter the differences for length (ΔLength X, ΔLength Z) and radius/diameter (ΔRadius/Δ∅) in the appropriate columns.

The wear data entered is added to the radius but subtracted from the tool length. A positive differential value for the radius therefore corresponds to an oversize (e.g. for subsequent grinding).

5.11 Activate tool monitoring

ShopTurn allows you to monitor the tool life of the tools automatically to ensure constant machining quality.

You can also disable tools that you no longer want to use or identify them as oversize.

The tool monitoring functions are activated via display machine data.

Machine manufacturer

Please follow the machine manufacturer's instructions!

Operating sequence

Tool wear

Select "Tool wear" via softkey

Tool life (T)

With the tool life T (Time), the service life for a tool with machining feedrate is monitored in minutes. When the remaining tool life is = 0, the tool is set to "disabled". The tool is not put into operation on the next tool change. If a replacement tool is available, it is inserted in its place.
Tool life is monitored on the basis of the selected tool cutting edge.

Count (C)	With the count C, the number of workpieces machined by the tool is counted. The tool is also disabled in this case, when the remainder reaches "0".
Wear (W)	With wear W, the greatest value in the wear parameters Δ Length X, Δ Length Z or Δ Radius or Δ ∅ in the wear list is monitored. Here too the tool is disabled if one of the wear parameters reaches the value for wear W.

Machine manufacturer Please follow the machine manufacturer's instructions!

Prewarning limit	The pre-warning limit specifies a tool life or quantity at which an initial warning is output.
Disabled (G)	Individual tools can also be disabled manually if you no longer want to use them for workpiece machining.
Oversize (U)	In the case of oversize tools, neighboring magazine locations are only reserved alternately, i.e. you can only insert the next tool in the next magazine location but one (this can also contain an oversize tool).
Coded for fixed location (P)	You can assign tools a fixed location, i.e., the tool can only be used at the current magazine location. The tool returns to the old magazine location when switching back.

Monitor tool use

Tool wear — Select "Tool wear" via softkey

▲ ▼ — Position the cursor on the tool that you want to monitor.

In column "T/C", select option "T" if you want to monitor the tool life. (T = Tool life, C = Count, W = Wear).

Enter a pre-warning limit for tool life, count or wear in minutes.

Enter the scheduled service life for the tool, the scheduled number of workpieces to be machined or the maximum permissible wear.

The tool is disabled when the tool life, count or wear is reached.

If you wish to monitor the count, you must also insert the following G code commands before the end of the program in every program that calls the tools to be monitored:

```
SETPIECE(1)          ; increase count by 1
SETPIECE(0)          ; delete T no.
```

Enter tool status

Option G

Place the cursor on a tool.

Select the option "G" in the first field of the last column if you want to disable the tool for machining.

-or-

Option U

Select the option "U" in the second field of the last column if you want to mark the tool as oversize.

The tool disable or location disable for neighboring magazine locations is now active.

5.5.12 Manage magazine locations

The magazine locations are listed with their tools, magazine locations are indicated as disabled/not disabled, and the properties assigned to the active tool (e.g. oversize) are displayed in the magazine list.

Disable magazine location

If a magazine location is defective, or when an oversize tool requires more than half a neighboring location, you can disable the magazine location.

Operating sequence

Magazine

Select with the "Magazine" softkey

Place the cursor on the relevant empty magazine location in the "Location disable" column.

Use the "Alternative" softkey to toggle the setting until a "G" (=disabled) appears in the field.

The location disable is now active and you can no longer assign tool data to this magazine location.

Enable magazine location

Place the cursor on an empty magazine location in the "Location disable" column.

Alternative

Press the "Alternative" softkey until the letter "G" no longer appears.

The magazine location is enabled again.

5.6 R parameters

5.6.1 Function

Function

Parameters are read and written by programs.
In this operating area, parameters can be altered manually.

5.6.2 Editing/deleting/finding R variables

Function

The number of channel-specific R variables is defined in machine data.

Range:
R0–999 (dependent on machine data).
There are no gaps in the numbering within the range.

Operating sequence

R variables

The "R variables" window appears.
The channel-specific parameters are displayed.
The vertical softkey bar changes.

Change parameters:
Position the cursor bar on the appropriate input field and enter the new values.

Delete parameters:

Delete area

Displays a marker in which the Rx to Ry parameter range to be deleted must be entered.

Delete all | OK

The complete R variable range is deleted after a safety query and "OK", i.e. all values are set to 0.

Abort

You cannot delete using "Abort".

Find

Finding parameters:
An input window for a parameter number appears when you press the "Find" softkey.

Enter the R variable number you wish to find via the numeric keypad. When you press the "Input key", the cursor is automatically positioned on this parameter if it exists.

Further information

Input and deletion of parameters can be disabled via the key switch.

5.7 Setting data

5.7.1 Working area limitation

Function

The "Working area limitation" function can be used to limit the range within which a tool can traverse in all channel axes. These commands allow you to set up protection zones in the working area which are out of bounds for tool movements.

Operating sequence

Setting data

Select softkey "Setting data".
The vertical softkey bar changes.

Working area limitat.

Select softkey "Working area limitation".
The "Working area limitation" window opens.

◀ ▶

Alter working area limitation:
Position the cursor on the desired field.
Enter the new values on the numeric keypad.
The upper or lower limit of the protection zone changes according to your input.

SELECT

Activate the appropriate working area limitation with the "Select key".

In "MDA" and "Automatic" modes, the working area limitation is not activated according to setting data within the current NC program until a "WALIMON" command is set.

Further information

The "Working area limitation" function can be disabled by means of the key switch.

5.7.2 Jog data

Function

The feedrates must be specified in the unit determined by the G function.

G function

G94 Feedrate in mm (inch)/min
G95 Rotational feedrate in mm (inch)/rev

Jogfeedrate

Feedrate value in Jog mode

Jogcontinuous

- Jog mode: Axis moves as long as key is pressed.
- Continuous mode: Axis moves after pressing key once, until
  - the key is pressed again,
  - NC Stop,
  - Reset,
  - SW/HW limit switches.

Variable increment

Increment value for Jog variable increment

The following data are displayed only if a spindle is configured:

Jog spindle speed

Spindle speed in Jog mode

Spindle

Jog data for the master spindle:

- Spindle no. Name of master spindle
- Direction of rotation: Direction of rotation of master spindle
- Spindle speed: Speed of the master spindle in Jog mode

Operating sequence

Setting data

Select softkey "Setting data".
The vertical softkey bar changes.

Jog data

Select softkey "Jog data".
The "Jog data" window is opened.

Change jog data:
Position the cursor bar on the appropriate input field and enter a new value or

SELECT

Select a new value using the "Select" key.

Further information

The limit values for the maximum and minimum permissible values are defined in the machine data.

5.7.3 Spindle data

Function

Max./min.

The value entered for the spindle speed in the fields max./min. must be within the limit values defined in the machine data. In addition, a further spindle speed limitation can be active in the program on account of the SIMS command.

Operating sequence

Setting data

Select softkey "Setting data".
The vertical softkey bar changes.

Spindle data

Select softkey "Spindle data".
The "Spindle data" window is opened.

Change spindle data:
Position the cursor bar on the appropriate input field and enter a new value or

select a new value using the "Select" key.

SELECT

Further information

- The limit values for the maximum and minimum permissible values are defined in the machine data.
- The "Spindle data" function is displayed only if a spindle is configured.

5.7.4 Dry run feedrate for DRY mode

Function

The feedrate entered here is used in the active program instead of the programmed feedrate when the function "Dry run feedrate" (program control) is selected in "Automatic" mode.

Operating sequence

Setting data

Select softkey "Setting data".
The vertical softkey bar changes.

Feedrate DRY

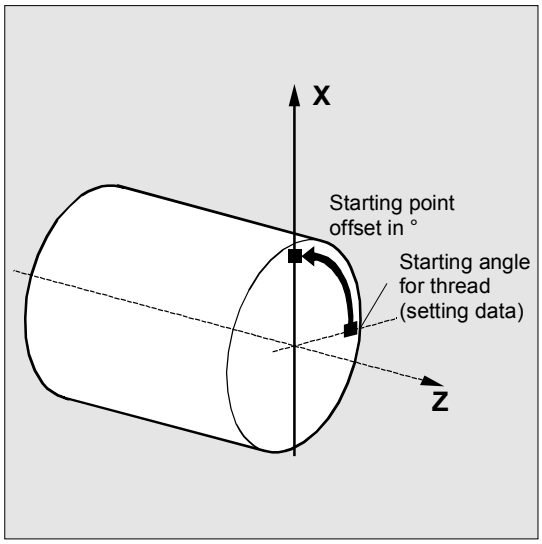
Select "Feedrate DRY" softkey.
The "Dry run feedrate" window is opened.

Change the dry run feedrate:
Enter a new value.

5.7.5 Starting angle for thread cutting

Function

For thread cutting, a starting position for the master spindle is displayed as the starting angle. A multiple thread can be cut by changing the angle when the thread cutting operation is repeated.



Operating sequence

Setting data

Select softkey "Setting data".
The vertical softkey bar changes.

Starting angle

Change starting angle:
Select softkey "Starting angle".
The "Starting angle for thread" window opens.

Enter a new value.

5.7.6 Other types of setting data

Function

All the setting data in the control are displayed in tabular form sorted according to general (i.e. NCK-specific), channel-specific and axis-specific setting data. The table contains both the setting data on the vertical softkeys such as working area limitation, Jog data etc., as well as special setting data such as software cam, oscillation, compensation etc.

Operating sequence

Setting data

Select softkey "Setting data".
The vertical softkey bar changes.

Display setting data:

Misc.

Select softkey "Misc.".
The horizontal and vertical softkey bars change.
Select the type:

General SD

- The "General setting data ($SN_)" window is opened.

Channel-specific SD

- The "Channel-specific setting data ($SC_)" window is opened.

Axis-specific SD

- The "Axis-specific setting data ($SA_)" window is opened.

The current setting data of the corresponding type $SN_, $SC_or $SA_ are displayed.

Find setting data:

Find

In the "Find setting data" window enter the name or number you are looking for (initial identifier is enough).

Find next

If several setting data have the same initial identifier, you can display other setting data by selecting softkey "Find next".

Change setting data:
Position the cursor bar on the appropriate input field and enter a new value.

Further information

Data can be edited or not depending on the active access protection level.

5.7.7 Protection zones

Function

The "Protection zones" function allows you to protect various elements on the machine, your equipment or the machined workpiece against incorrect axis motions. You can view up to 10 programmed protection zones in levels G17, G18 and G19.

References /PGA/ Programming Guide, Advanced

Operating sequence

Setting data

Protection zones

Select softkey "Setting data".
The vertical softkey bar changes.
Select softkey "Protection zones".
The "Working area limitations and protection zones" window opens.
The vertical softkey bar changes again.

Protection zone +

Protection zone –

Select softkey "Protection zone +" or "Protection zone -".
Up to 10 protection zones are displayed in succession.

Select the plane in which the relevant protection zone is located:

G17

- Plane G17 (X,Y; positioning direction Z)

G18

- Plane G18 (Z,X; positioning direction Y)

G19

- Plane G19 (Y,Z; positioning direction X)

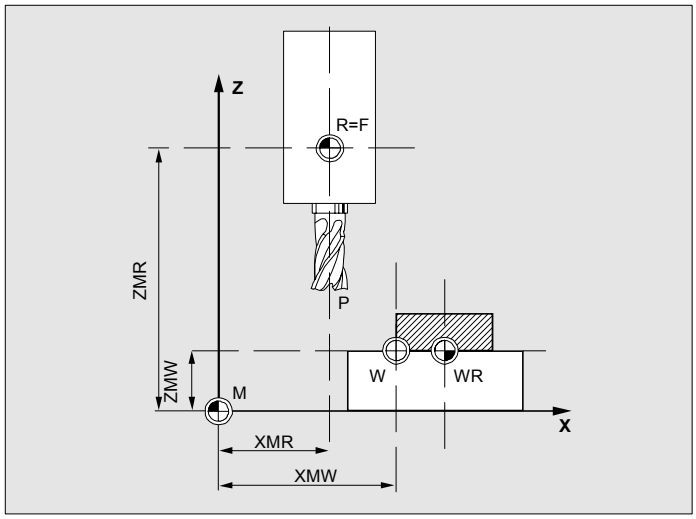
5.8 Zero offset

5.8.1 Function

Machine/tool zero

The actual values are referred to the machine zero after reference point approach. The machining program of the workpiece refers to the workpiece zero.
The machine zero and workpiece zero are not necessarily identical. Depending on the type of workpiece and the way it is clamped, the distance between the machine zero and workpiece zero can vary. In part program processing this is compensated for by the work offset.

Work offset on a milling machine



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

Effective WO

The work offset effective in an axis
`$P_ACTFRAME=..` is calculated from the **sum** of the following zero offsets:

Settable WO

You can activate a settable zero offset in the program you have called with G54 to G57 and other G functions or with `$P_IFRAME=...`
Basic zero offset (basic frame): Is displayed like a settable WO.

Programmable WO

You can use the programmable zero offset `$P_PFRAME=..` to program an additional work offset for geometry and special axes in the part program you have called.
The values of the programmed work offsets are deleted with end of program or reset.

External WO

In addition to all the offsets which define the position of the workpiece zero point, an external zero offset can be overlaid by means of the handwheel (DRF offset) or from the PLC.

DRF offset

Differential Resolver Function: An NC function which generates an incremental zero offset in automatic mode in conjunction with an electronic handwheel.

Frame

Frame is the conventional term for a geometrical expression that describes an arithmetic rule, such as translation or rotation.
Frames are used to describe the position of a destination coordinate system by specifying coordinates or angles starting from the current workpiece coordinate system.

Possible frames

- Basic frame (basic offset)
- Settable frames (G54...G599)
- Programmable frames

References

/PGA/ Programming Guide Advanced

Frame components

Frame components
A frame can consist of the following arithmetic rules:

- Zero point offset, TRANS, ATRANS
- Rotation, ROT, AROT
- Scale, SCALE, ASCALE
- Mirroring, MIRROR, AMIRROR

In the part program, all work offsets can be deselected non-modally with G53.

5.8.2 Display zero/work offsets

Function

In the overview, all existing settable work offsets are listed. The number of possible work offsets is defined by a machine datum.
The first settable work offsets G54 to G57 are permanently assigned the identifiers `$P_UIFR[1]` to `$P_UIFR[4]`.

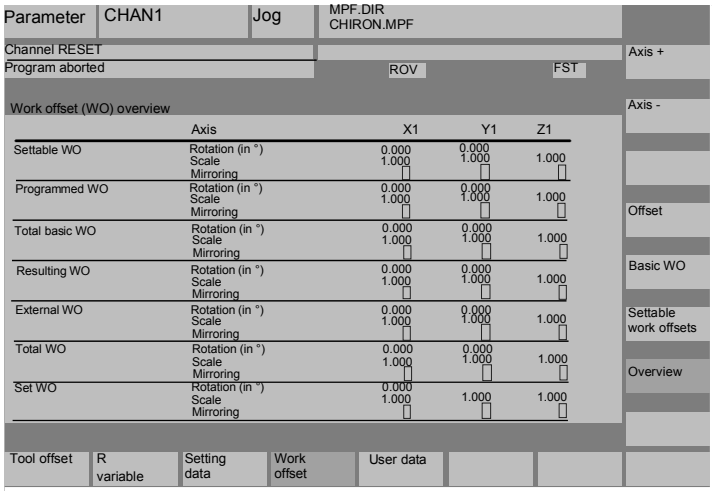
Operating sequence

Zero offset

Press the "Zero offset" softkey.
The vertical softkey bar changes.

Overview

Select the "Overview" softkey and the following overview appears:



Select work offsets:

Axis +

- The display switches to the defined work offsets of the next axis.

Axis -

- The display switches to the defined work offsets of the previous axis.

Offsets | Rotation scal., mirr.

You can use these softkeys to change the display mode of the currently displayed work offsets.
For example:

- either the absolute offsets (coarse and fine) with reference to the coordinate axes
- or the individual values split into the components rotation, scaling and mirroring.

You can select and, if necessary, change the individual values of the work offsets in both display modes.

Display other work offsets:

Basic WO

All defined basic work offsets (global and channel-specific) are displayed in a table.

Settable work offsets

All defined settable work offsets are displayed in a table where they can be edited if necessary (select and edit).

5.8.3 Changing the settable work/zero offset (G54 ...)

Function

$P_UIFR[]

This identifier can be used to change a settable zero offset in the program.

Coarse offset

The value of the coarse offset is defined for the relevant axis.

Fine offset

The data limits (absolute) are set for the fine work offset via the machine data. The fine offset is displayed in the "Settable work offset" screen.
The work offset is activated via MD.

Machine manufacturer

The basic work offset is activated by MD.
Please follow the machine tool manufacturer's instructions!

Rotation

The value of the rotation around the respective geometry axis (e.g. X, Y, Z) can be entered.
Rotation can only be programmed around geometry axes.

Scale

The scale factor can be defined for the respective axis.

Mirroring

Mirroring of the relevant axis around the coordinate zero can be activated and deactivated.

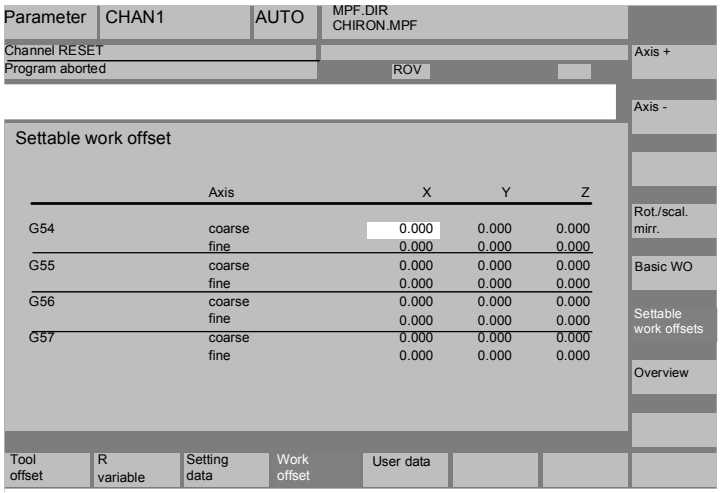
Operating sequence

Work offset

Select softkey "Work offset".
The vertical softkey bar changes.

Settable work offsets

The "Settable work offsets" window opens.



You can choose a work offset selectively from the overview of work offsets. Select the appropriate fields with the cursor

and overwrite the fields you want to change with the new value.

Select a new value via the "Select" key (with mirroring).

The work offsets are saved, i.e. transferred to the NC.

5.8.4 Activate work/zero offset and basic frame immediately

Function

The machine data can be set to specify that the work offset and basic frame can be activated immediately if the part program switches to the "Reset" state. This also occurs if the part program was first switched to JOG status.
If the channel is in the "Reset" state, active work offset and basic frame are not activated until the part program is continued.

Further information

When the function is used in the Reset state, the machine data must be set such that the settable work offset or basic frame is not reset when the program switches to Reset.

Machine manufacturer Please follow the machine tool manufacturer's instructions!

References /FB/ K2: Axis Types, Coordinate Systems, Frames

Danger

The offset is applied the next time the parts program is started.

5.8.5 Global work/zero offset/frame (Basic WO)

Function

In addition to the settable, the programmable and the external work offsets, it is possible to define up to 16 **global** work offsets/frames (basic WO). This allows offsets, scales and mirrors to be defined simultaneously for all channel and machine axes.
The global work offsets (NCU-global frames) apply uniformly to all channels. They can be read and written from all channels. The activation is performed in the relevant channel.

Basic WO (total basic frame)

In addition, 16 channel-specific basic work offsets can be defined in each channel. The global and channel-specific frames are combined to produce a total basic frame (basic WO).

Machine manufacturer

Recommendation:
Use the 3rd basic offset onwards for your own applications. The 1st and 2nd basic offsets are reserved for setting the actual value and the external work offset.

With global frames there is no geometrical relationship between the axes. It is therefore not possible to perform rotations or program geometry axis identifiers.
In SW 5 and higher, the settable work offset and the basic work offset are represented in **one** table. You can edit the values in this table. You can switch between the values of the individual axes.

For **all** work offsets, you can display either the defined offsets (coarse and fine) or the specified rotations, scales and mirrors for each value.

References

/FB/ K2: Axis Types, Coordinate Systems, Frames

Operating sequence

Work offset

Select softkey "Work offset".
The vertical softkey bar changes.

Basic WO

All defined basic work offsets (global and channel-specific) are displayed in a table.
The display mode can be changed by softkey (see above).
You can edit the values directly in the table.
Rotations are not possible with global frames, since no geometrical relationship exists between the axes in this case.

Further information

The work offset must be changed only when the NC program is stopped. Changes are updated immediately. The work offset values in the display are updated cyclically.

5.9 User data/user variables (GUD, PUD, LUD)

5.9.1 General

Function

User data (UD) can be defined by means of a variety of variables:

- GUD – global variables which are valid in all programs.
- LUD – local variables which are valid only in the program or subroutine in which they have been defined.
- PUD-Program-global user data.

The display of global user data (GUD) can be locked by means of the keyswitch or a password.

5.9.2 Changing/finding user data/user variables

Operating sequence

Press softkey "User data".
The "Global user data" window is displayed.
The vertical softkey bar changes.
You can toggle between windows

Global user data

- "Global user data" (GUD)

Channel-spec. u. d.

- "Channel specific user data" and

Program user data | Local user data

- "Program user data" or "Local user data"

Program-global variables (PUD) and local variables (LUD) are displayed.

PAGE DOWN or PAGE UP

You can scroll up and down in the list using the "Page" keys.

Changing user data

Position the cursor on the user data that you wish to edit and enter a new value or

Select a new value using the "Select" key.
New values are automatically accepted.

Searching for user data

GUD + GUD -

Press softkeys "GUD +" and "GUD - " to scroll through user data from GUD 1 to GUD 9.

GUD:

The "Select global user data" window is opened. The following values are permissible:

- 1 = SGUD (Siemens)
- 2 = MGUD (Machine manufacturer)
- 3 = UGUD (Machine user data)
- 4 ... 9 = GD4 ... GD9 (other, e.g. grinding cycles, etc.)

The selected data are displayed in the "Global user data" window.

Find

Select "Find" softkey.
Dialog window "Find user data" appears on the screen.

The data name or a character string within the name can be entered as the search target. The cursor must be positioned on the user data to be found.

Find next

The next user data with the initial identifier searched for is displayed.

User data of types `AXIS` and `FRAME` are not displayed.
Only those local user data that still exist in the execution chain of the control are displayed.

The list of local user data for the display is updated on every "Cycle Stop", but the values are updated continuously.
Before global user data definitions can be made operative in the control, it may be necessary to set machine data.

Further information

How to define and activate user data is described in
Chapter 6: Program operating Area.

■

Program Operating Area

6.1 Program types

6.1.1 Part program

A part program consists of a sequence of instructions to the NC control. In its entirety, this sequence affects the production of a specific workpiece or a particular machining process on a given blank.

6.1.2 Subprogram

A subprogram is a sequence of instructions in a part program which can be called repeatedly with different defining parameters. Cycles are a form of subprogram.

6.1.3 Workpiece

- A workpiece is a part to be produced/machined by the machine tool.
- In the HMI, a workpiece is a directory in which programs and other data for machining a particular workpiece are stored.

6.1.4 Cycles

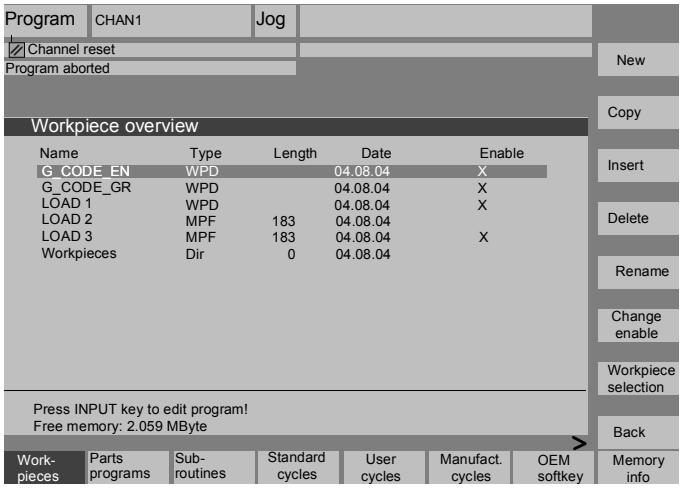
Cycles are subprograms for the execution of a recurring machining process on the workpiece.

6.2 Storing of programs

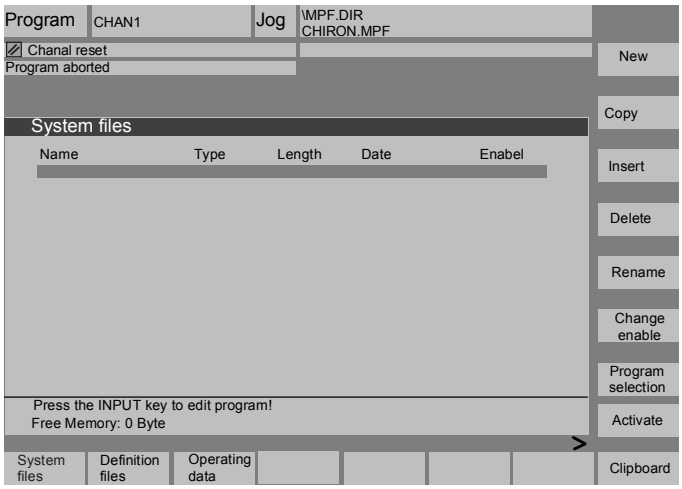
Programs are saved in the NC memory. The size of this memory is dependent on settings made during start-up. (See Chapter 6 "Memory info")

6.3 Program basic display

The Program basic display contains a complete overview of all workpiece and program directories.



Additional softkeys are available via the Expand key:



Horizontal softkeys

Work-pieces

This key displays an overview of all the workpieces you have created with their part programs.

Parts programs

An overview of all part programs (main programs) stored in the selected directory is displayed.

Sub-routines (子程序)	An overview of all subprograms stored in the selected directory is displayed. Subprograms are processed in the same way as described for "Process main programs".
Disk, Network drive, Compact Flash Card	These four softkeys (logical drive 1–3 and the OEM softkey) appear only if the "network and disk drive management" option has been enabled, e.g. as "Disk", "Network drive", "Compact Flash Card" (see Chap. 9) or
OEM softkey	e.g. OEM softkey (expand operator interface)
Standard cycles	The standard cycles appear on the screen when you press the softkey "Standard cycles".
Manufact. cycles	Select softkey "Manufacturer cycles" to display a list of the cycles that have been integrated by the manufacturer.
User cycles	Select softkey "User cycles" to display a list of the user cycles that you have added.
Memory info	The total available/used memory is displayed.
>	You can access the following softkeys by pressing the ETC key:
System files	"System files" (e.g. OSTORE1_SYF),
Definition files	"Definition files" (e.g. GUD4_DEF),
Operating data	"Operating files" (e.g. BD_TEA).
Clipboard	By pressing the softkey "Clipboard" you can display a list of the programs/data stored in the clipboard.

Machine manufacturer

These softkeys can be disabled in display machine data.
Please note information supplied by machine tool manufacturer!

Vertical softkeys

New	Creates a new file for a workpiece/part program.
Copy	The current file name including the directory is marked and copied to the clipboard. If you delete the copied file from the clipboard, you will not be able to paste it again.
Insert	Pastes a file copied to the clipboard into the current directory; the file name must be altered and confirmed beforehand.
Delete	Deletes a file (workpiece/part program).
Rename	Overwrites the file name/type of a workpiece/part program.
Change enable	Sets/resets the enable for a workpiece/part program. The enable must be set (X) for a workpiece/part program to be selected.
Workpiece selection / Program selection	Selects a workpiece/part program for execution in the currently active channel.
Back	When you branch to a workpiece, the "Back" softkey is displayed to allow you to return to the workpiece overview.

6.4 Edit programs

6.4.1 Text editor

The ASCII editor provides you with the following functions:

- Switch between insert and overwrite mode.
- Mark, copy, delete block.
- Paste block.
- Position cursor/find/replace text.
- Create contour (programming support).
- Configure cycle parameters (drilling, milling, turning).
- Start simulation.
- Recompile (cycles, free contour programming).
- Renumber blocks.
- Change settings.

For a detailed description, please refer to:
Chapter 2, "General operating sequences".

6.4.2 Selective program protection: RO

Function

In programs written with program templates or when using the programming support functions, certain machine-specific code lines may be protected against changes.

A read-only identifier (";*RO*") is tagged onto the code blocks as a comment. The ASCII editor recognizes these blocks, hides them or displays them in the read-only text color (gray), and prevents changes to these blocks.

The read-only identifier (";*RO*") shows you which part of the program is protected.
Any attempt to change a program part protected by the read-only identifier is denied with the message "Block cannot be written."

Further information

When creating a program template, please make sure that the read-only identifier appears immediately at the end of the block.

6.4.3 Hidden program lines: Display HD

Function

To display hidden, write-protected text (with the identifier;*HD) in the editor, press the Etc key and the "Settings" softkey.

Machine manufacturer

This function must be set up by the machine manufacturer. Please note the information supplied by the machine tool manufacturer!

6.4.4 Reserved character string

Function

Reserved character strings occur in part program code lines that arise by calling cycles and contour programming. They become visible when activated if the "Display hidden lines" setting is active in the editor.

The following character strings must not be used in part program lines that are input directly:
;#
;#END
;NCG
;*RO*
;*HD*

Further information

See Section: Selective Program Protection RO
Hidden program lines: display HD

6.4.5 Defining and activating user data (GUD, LUD)

Function

Defining user data (GUD)

By editing a DEF/MAC file, you can alter or delete existing definition macro files or add new ones.

Definition files

In the "Program" operating area, select the "ETC key" and then "Definition files"; an overview of the files appears.

New
Copy
Delete

You can alter the definitions by selecting the "New," "Copy" and "Delete" softkeys. These alterations are stored in the current file.

Rename

You can create a new definition file, e.g. from a backup file (SGUD.BAK), by selecting the "Rename" softkey and changing the extension.

Activating user data (GUD)

Change enable

To activate an edited definition file, select the "Change enable" softkey.

Activate definition file:
Position the cursor on the definition file and press the "Activate" softkey.
The following message appears in the dialog line:
"Do you want to activate the definitions in this file?"

Activate

No

"No" The modified data remains in the file. The changes are not saved.

Yes

"Yes" The modified data is activated.
Another question is displayed: "Should the previous definition data be retained?"

Yes

"Yes" The screen form is closed, the definitions are activated and the previous definition data is retained.

No

"No" The screen form is closed, the definitions are activated and assigned to the file with the predefined values.

Error messages:
The following error messages appear in the dialog line:
"Error occurred during activation of file"

Output of NC alarms:
Once the maximum number of files on the NC has been reached, no more backup files can be created. The following acknowledgeable NC alarms appear: "Too many part programs in the NC memory"
"NC memory full"

The same applies when activating macro files (.MAC).

6.4.6 Defining and activating user data (GUD, LUD) via RS-232 C

Function

1. Save `_N_INITIAL_INI` block via RS-232 C.
Create a definition file for user data in the Program operating area:

- Predefined file names are used:
  `_N_GUD1_DEF` (global Siemens data),
  `_N_GUD2_DEF` (global machine manufacturer data),
  `_N_GUD3_DEF` (global user data),
  `_N_GUD4_DEF` to `_N_GUD9_DEF` (additional global data, e.g. grinding cycles, etc.)
- Files with these names can contain definitions of GUD variables. The same rules apply to these as to LUD variable definitions.

2. Load definition file to main memory via RS-232 C.
The control system always creates a directory named `_N_DEF_DIR`. This name is entered in the header of the GUD definition file as a path:
Example:

```
_N_GUD1_DEF
$PATH=/_N_DEF_DIR
DEF NCK REAL NCKVAR
DEF CHAN INT CHANVAR
M17
```

3. Load "Init files": User definition files are active.

4. Data backup
The modified GUD data contents are saved when block _N_INITIAL_INI is read out via "Data out" in the Services operating area. These data can only be reloaded into the control if it has first been ensured that the required definition files are in the control.
Definition and creation of user data

Data is only saved for variable definition files, not for macros.

References /PGA/Programming Guide, Advanced

6.5 Free contour programming

6.5.1 General

Function

The free contour programming is a support tool for the editor.
The contour programming function enables you to create simple or complex contours.
An integrated contour calculator calculates any missing parameters for you, provided that they can be computed from other parameters.
A contour comprises separate contour elements, whereby at least two and up to 250 elements result in a defined contour. You can also program undercut, radii, chamfer or tangential transitions between the contour elements.
The programmed contours are transferred to the edited part program.
The following contour elements are available for the definition of a contour:

- Vertical line

- Straight line (planar, longitudinal, inclined) horizontal

- Diagonal line

- Arc/circle

Further information

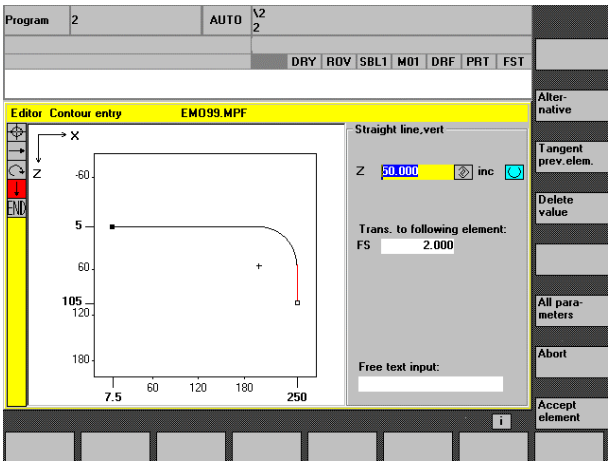
1. The valid geometry axes in the first channel are determined and used in the part program.
2. The contour elements of the contour chain are displayed by symbols or text. The mode of representation can be set via the "Setting contour" function in the editor.

Setting contour

6.5.2 Graphic representation of the contour

Function

The graphics window displays the progress of the contour chain as you parameterize the contour elements.
The currently selected element is displayed in orange in the graphics window.



The created contour element can be displayed in various line types and colors depending on its status:

HMI Embedded	Meaning
Black	Programmed contour
Orange	Current contour element
Yellow	Alternative element
Black continuous line	Defined element
Dotted line	Partially defined element
Dashed line	Alternative element

The current status of the contour is displayed insofar as it can be interpreted by the control on the basis of parameter inputs. If the contour is still not displayed in the programming graphic, further values still need to be entered. Check the contour elements you have already programmed. You may have forgotten to enter all the data you know.

The coordinate system scaling is automatically adapted to changes in the complete contour.

The position of the coordinate system is displayed in the graphics window.

6.5.3 Create contour

Function

For each contour that you want to cut, you must create a new contour. The first step in creating a contour is to specify a starting point.
You have the option of beginning the contour with a transition element to the blank. You can also enter any additional commands (up to 40 characters) in G code format for the start point.
If you want to create a contour that is similar to an existing contour, you can copy the existing one, rename it and just alter selected contour elements. However, if you want to use an identical contour at another place in the program, you must not rename the copy. Changes to the one contour will then automatically be applied to the other contour with the same name.

Sequence

Work- pieces | Part program

To select an existing program via the "Workpiece" and "Part program" softkeys or create a new part program with the "New" softkey.
Enter a name and confirm with "OK".
You are now back in the ASCII editor.
You can access the following softkeys by pressing the ETC key:

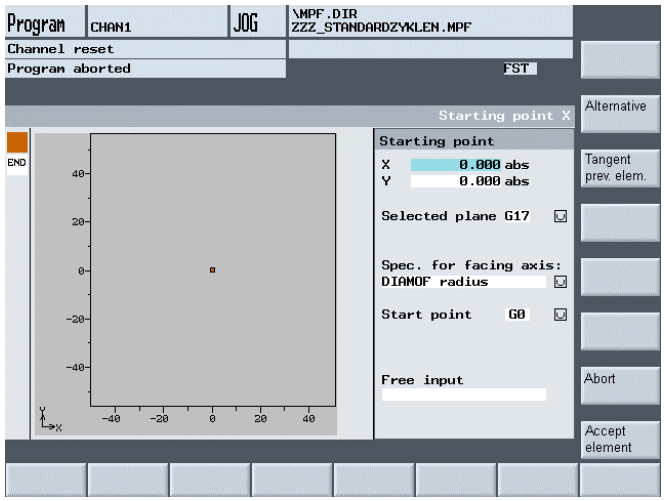
>

Support | New contour

Open the contour editor by pressing the "Support" and "New contour" softkeys.

Define the start point

The input form for the start point of the contour appears.



When entering a contour, begin at a position which you know and enter it as the starting point. You can enter Cartesian or polar coordinates.

To define the geometry axes, choose from the planes G17, G18 and G19 with the Select Key in the field behind "Plane selection". The coordinate system changes accordingly.
The default tool axis (defined in the machine data) can be changed for machines with more than two axes. The associated start point axes are automatically adjusted.

Alter-native

Position the cursor on the "Facing axis dimension" field and click on the field using softkey "Alternative" (or with the "Selection" key) repeatedly until the dimension you require is displayed.

The approach motion to the starting point can now be changed from G0 (rapid traverse) to G1 (linear interpolation) via the new field "Approach starting point".

You can define a specific feed rate for G1 via the "Free text input" field,
e.g. `G95 F0,3.`

Accept element

Select softkey "Accept element" to store the starting point.

Abort

With the "Abort" softkey, your settings are discarded and you arrive at the previous screen form.

Cartesian start point

Select the machining plane.

Enter the starting point for the contour.

Enter any additional commands in G code format, as required.

Accept element

Press the "Accept element" softkey.

Enter the individual contour elements (see Section "Create contour elements").

Polar start point

Select the machining plane.

Pole

Press the "Pole" softkey.

Enter the starting point for the contour in polar coordinates.

Enter any additional commands in G code format, as required.

Accept element

Press the "Accept element" softkey.

Enter the individual contour elements (see Section "Create contour elements").

Close the contour

A contour always has to be closed.
If you do not wish to create all contour elements from starting point to starting point, you can close the contour from the current position to the starting point.

Close contour

Press the "Close contour" softkey.

A line from the current point to the start point is created.

Recompile contour

Recompile

You can edit an existing contour by selecting the softkey "Recompile". The editor cursor must be positioned inside the contour first.

Notice

During recompiling, only the contour elements that have been created with free contour programming are created again. In addition, only the texts that were added via the input field "free text input" are recompiled. Any changes you made directly in the program text are lost. However, you can still insert and edit free texts afterwards, which will not be lost.

Further information

The NC code generated by the contour programming in the part program must never be altered manually. Otherwise recompilation is no longer possible.
Exception: Insertion of block numbers and masking blocks.

Saving a contour element

If all contour elements and transition elements have been generated, save the contour by pressing the "Accept" softkey

Accept

6.5.4 Changing a contour

You can change a previously created contour later.

Individual contour elements can be

- appended
- changed
- deleted

If your program contains two contours of the same name, changes to the one contour are automatically applied to the second contour with the same name.

Sequence

Alter a contour element

Select the contour.

Press the "Cursor Right" key.

Position the cursor on the contour element that you want to modify.

Press "INPUT" key

The associated input form is opened and an enlarged view of the selected element appears in the programming graphic.

Accept element

After inputting the changes, press the "Accept element" softkey.

6.5.5 Contour elements general

Function

Contour chain

The elements of the contour are displayed symbolically in the sequence in which they were programmed in a contour chain next to the graphic window.

Symbolic representation

Contour element	Abbreviation	Symbol	Meaning
Start point	SP		Start point of contour
Straight line to the left	SL		Straight lines in 90° grid
Right	SR		Straight lines in 90° grid
Left/right	SLR		Straight lines in 90° grid
Top	SU		Straight lines in 90° grid
Bottom	SD		Straight lines in 90° grid
Above/below	SUD		Straight lines in 90° grid
Straight line in any direction	SA		Straight line with any pitch
Arc to the left	CL		Circle
Right	CR		Circle
Contour termination	END	END	End of contour

Color of symbols

The different color of the symbols indicates their status:

Foreground	Background	Meaning
-	Black	Cursor on new element
White	Black	Cursor on current element
Black	White	Normal (undefined) element
White	Black	Element currently detached (residual model)

6.5.6 Create, change, delete contour elements

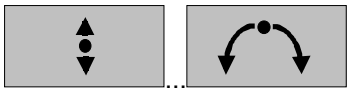
When you have created a new contour and specified the start point, you can define the individual elements that the contour comprises.

If you leave any parameter input fields blank, the control assumes that you do not know the right values and attempts to calculate them from the settings of other parameters.

The contour is always machined in the programmed direction.
As soon as you have entered an element, the input focus is moved to the contour chain on the left of the graphic display. The input focus has a yellow border. You can navigate within the contour chain using the cursor keys.

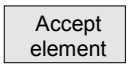
You can select an existing contour element with "INPUT". A new contour element is inserted after the cursor when you select one of the contour elements on the horizontal softkey menu; the input focus is then switched to the parameter input on the right of the graphic display. You can navigate around the contour chain again after selecting "Accept element" or "Abort". The following contour elements (example for turning: G18) are available for the definition of contours.

Entering a contour element



Select a contour element via softkey.

Enter all the data available from the workpiece drawing in the input form (e.g. length of straight line, target position, transition to next element, angle of lead, etc.).



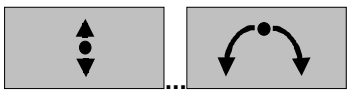
Press the "Accept element" softkey to accept all values.

The contour element is added to the contour. Repeat the procedure until the contour is complete.

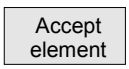
Append a contour element

Select the contour.

The individual contour elements are listed. Use the cursor keys to select the element in front of the end marker.



Select the required contour element via softkey.



Press the "Accept element" softkey.

The required contour element is appended to the contour.

Insert contour element

Use the cursor keys to select the contour element **behind** which you wish to insert another element.

Selecting a contour element

Position the cursor on the desired contour element in the contour chain, and select it with the "Input" key.
The parameters for the selected element will then be displayed.
The name of the element appears at the top of the parameterization window.
Once the contour element can be displayed geometrically, it is highlighted accordingly in the graphic display area, i.e. its color changes from black to orange.

Displaying additional parameters

All parameters

Alter-native

If your drawing contains further data (dimensions) for a contour element, select softkey "All parameters" to extend the range of input options for the element.

The softkey "Alternative" is displayed only in cases where the cursor is positioned on an input field with several selectable settings.

Define pole

Continue Pole

Accept element

If you wish to enter the contour elements Diagonal line and Circle/arc in polar coordinates, you must first define a pole.

Press the "Continue" and "Pole" softkeys.

Enter the coordinates of the pole.

Press the "Accept element" softkey.

The pole is defined. You can now choose between "Cartesian" and "Polar" in the input screen form for the Diagonal line and Circle/Arc contour elements.

Tangent to preceding element

Tangent to prec. elem.

When entering data for a contour element you can program the transition to the preceding element as a tangent.

Press the "Tangent to prec. elem." softkey.

The angle to the preceding element $\alpha 2$ is set to 0°. The "tangential" selection appears in the parameter input field.

Make a dialog selection

Select dialog

Select dialog Accept dialog

Some constellations of parameters can produce several different contour characteristics. In such cases, you will be asked to select a dialog. When you select the "Select dialog" softkey, the existing selection options are listed in the graphic display area.
Select the appropriate dialog (black continuous line) with "Select dialog" softkey and confirm by selecting "Accept dialog" softkey.

Changing a selected dialog

Change selection

Select dialog | Accept dialog

If you want to change an existing dialog selection, you must select the contour element in which the dialog was originally chosen. Both alternatives are displayed again when you select softkey "Change selection".
You can select another dialog.

If the selection has become unnecessary as a result of other input values, you will no longer be requested to select a dialog!

Transition element at contour end

A transition element can be used whenever there is a point of intersection between two neighboring elements which can be calculated from the input values.
You can choose among Radius **R**, a chamfer **FS**, and an **undercut** (thread, thread DIN, Form E, or Form F) as a transition element between any two contour elements. The transition is always appended to the end of a contour element. You select transition elements in the parameter input screen form for the relevant contour element.

Place the cursor on the last contour element.
Press the "Cursor Right" key.
The associated input screen form opens.

Enter a transition element.

Accept element

Press the "Accept element" softkey.

Abort

When you select "Abort", the contour element values are discarded and you return to the basic display. The input focus switches back to the contour chain.

Delete value

The values of the parameter are deleted.

Delete a contour element

Select the contour.

Press the "Cursor Right" key.

The individual contour elements are listed.

Delete element ... OK

Place the cursor on the contour element to be deleted.
Press the "Delete element" softkey and confirm with "OK".

The contour element is deleted.

Accept

To save the contour, press the "Accept" softkey.

Saving a contour element

Accept element

If you have entered the available data for a contour element or selected the desired contour by means of softkey "Select dialog", select softkey "Accept element" to store the contour element and return to the basic display.
You can then program the next contour element.

Further information

The NC code generated by the contour programming in the part program must never be altered manually. Otherwise recompilation is no longer possible.
Exception: Insertion of block numbers and masking blocks.

Parameters on gray background

These parameters have been calculated by the control system. You cannot alter them.
When the programmed parameter input fields (white background) are altered, the control calculates new data, which are then immediately displayed again in the input screen form.

Input value is already calculated

With some contours, the control system may already have calculated an input value from other settings.
Problems may then arise if the control-calculated value does not tally with the workshop drawing. In this case, you must delete the settings from which the control has automatically calculated the input value. You can then enter the setting exactly from the workshop drawing.

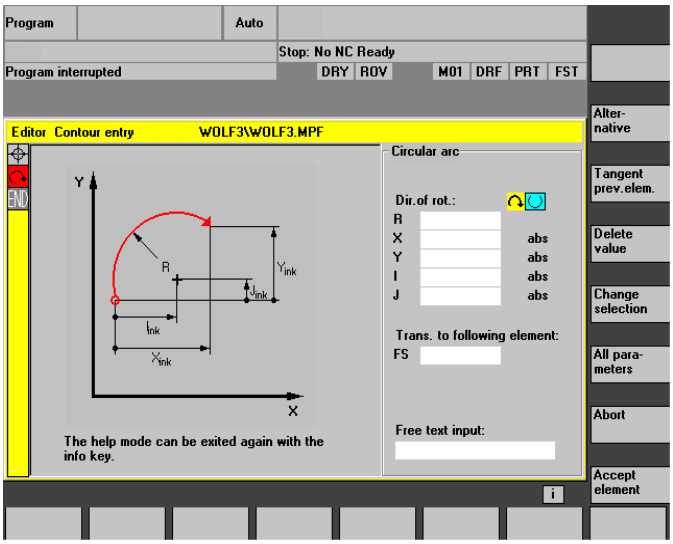
Settings

The technology (turning/milling) and the position of the coordinate system are read from the appropriate machine data. You can see the selected configuration with "Settings".

6.5.7 Help

HELP

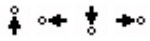
When you enter parameters you can call up a help screen with the Info key which graphically represents the parameters you are entering. The help screen that appears depends on the cursor position in the parameter display.
The help screen is displayed on top of the parameter screen.

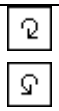


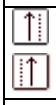
If you press the Info key again the help screen is closed and the display graphic is activated again. The help screens displayed correspond to the selected coordinate system. The axis names are derived from the current geometry axis names.
Help screens are displayed for the following entries:

- Start point
- Vertical line
- Straight vertical line, angle entry field
- Horizontal line
- Straight horizontal line, angle entry field
- Straight line in any direction
- Straight line any, angle entry field
- Circle
- Circle, angle entry field
- Radius/chamfer

6.5.8 Parameter description of the contour elements line/circle and pole

Parameters	Contour element "Straight line"	Unit
X absolute X incremental	Absolute end position in X direction Incremental end position in X direction	mm mm
Y absolute Y incremental	Absolute end position in Y direction Incremental end position in Y direction	
L	Length of line	mm
α1	Pitch angle referred to X axis	Degr.
α2	Angle to preceding element; tangential transition: α2=0	Degr.
FB	Feedrate for contour element "Straight line"	mm/rev
Transition at contour start	FS: Chamfer as transition element at contour start R: Radius as transition element at contour start FS=0 or R=0: No transition element	mm mm
	Location of transition element relative to contour start point 	
Undercut size	Undercut size acc. to DIN table (for forms E and F only): Radius/depth, e.g.: E1.0x0.4 (undercut form E) or F0.6x0.3 (undercut form F)	
FRC	Feedrate for transition element chamfer or radius	mm/rev
CA	Allowance for subsequent grinding	mm
	Grinding allowance to right of contour (viewed from starting point) Grinding allowance to left of contour (viewed from starting point)	
Additional command	Any additional command in G code format	

Parameters	Contour element "Circle"	Unit
Direction of rotation	 Clockwise rotation Counterclockwise rotation	
X absolute X incremental	Absolute end position in X direction Incremental end position in X direction	mm mm
Y absolute Y incremental	Absolute end position in Y direction Incremental end position in Y direction	
Z	Target position in the Z direction (abs. or inc.) Incremental dimensions: The plus/minus sign is evaluated.	mm
α1	Starting angle referred to X axis	Degr.
α2	Angle to preceding element; tangential transition: α2=0	Degr.
β1	End angle referred to X axis	Degr.
β2	Angle of aperture of circle	Degr.
FB	Feedrate for circle contour element	mm/rev
R	Radius of circle	mm
I	Position of arc center point in X direction (abs. or incr.)	mm
K	Position of circle center point in Z direction (abs. or inc.) Incremental dimensions: The plus/minus sign is evaluated.	mm

J	Position of arc center point in Y direction (abs. or incr.)	mm
Transition to following element	Transition element to next contour is a chamfer (FS) Transition element to next contour is a radius (R) FS=0 or R=0 means no transition element	mm mm
FRC	Feedrate for transition element chamfer or radius	mm/rev
CA	Allowance for subsequent grinding	mm
	Grinding allowance to right of contour (viewed from starting point) Grinding allowance to left of contour (viewed from starting point)	
Additional command	Any additional command in G code format	

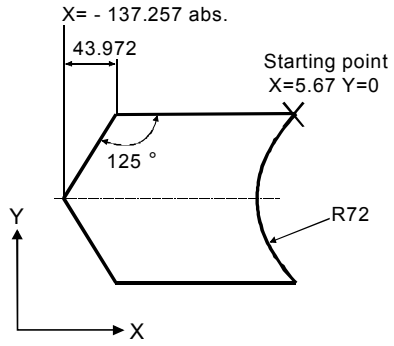
Machine manufacturer The names of the identifiers (X or Y ...) are defined in the machine data where they can also be changed.

6.5.9 Programming example of free contour programming

Example 1

Starting point: X=5.67 abs., Y=0 abs., machining plane G17
The contour is programmed in a counter-clockwise direction.

Workpiece drawing of contour

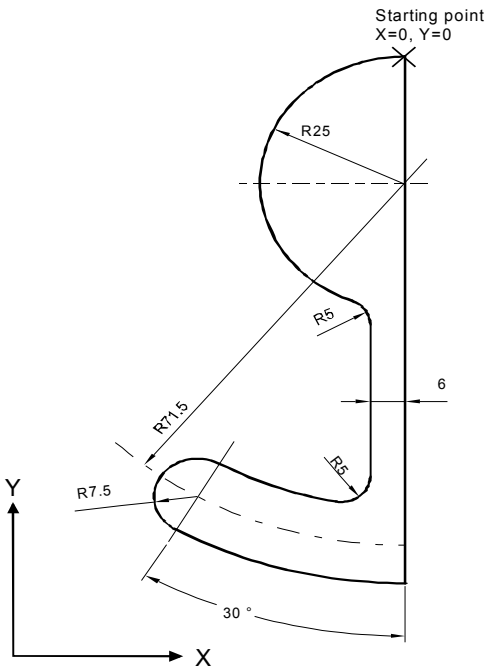


Element	Softkey	Parameters	Remarks
1		All parameters, α1=180 degrees	Observe angles in help screen!
2		X=–43.972 inc, all parameters X=–137.257 abs α1=–125 degrees	Definition of coordinates in X in "abs" and in "inc" Observe angles in help screen!
3		X=43.972 inc α1=–55 degrees	Definition of coordinates in X in "inc" Observe angles in help screen!
4		X=5.67 abs	
5		CW direction of rotation, R=72, X=5.67 abs., Y=0 abs., Make a dialog selection	

Example 2

Starting point: X=0 abs., Y=0 abs. , machining plane G17
The contour is programmed in the clockwise direction with dialog selection.
For this contour it is advisable to display all parameters via the softkey "All parameters".

Workpiece drawing of contour

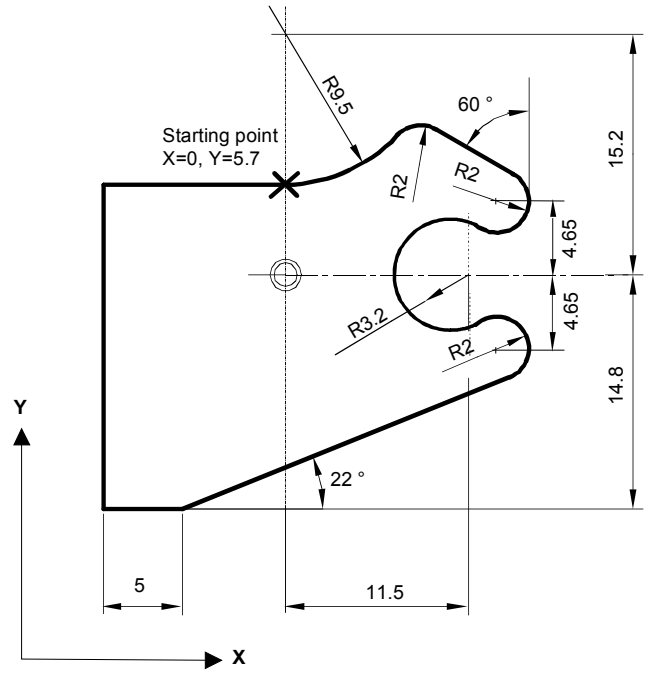


Element	Softkey	Parameters	Remarks
1		Y=–104 abs.	
2		Direction of rotation right, R=79, I=0 abs., Select dialog, all parameters, β2=30 degrees	
3		CW direction of rotation, tangent to preced. R=7.5, all parameters, β2=180 degrees	
4		CCW direction of rotation, R = 64, X =–6 abs., I=0 abs., Make dialog selection, make dialog selection Transition to following element: R=5	
5		All parameters, α1=90 degrees, Transition to following element: R=5	Observe angles in help screen!
6		Direction of rotation right, R=25, X=0 abs., Y=0 abs. I=0 abs Make dialog selection, make dialog selection.	

Example 3

Starting point: X=0 abs., Y=5.7 abs., machining plane G17
The contour is programmed in a clockwise direction.

Workpiece drawing of contour



Element	Softkey	Parameters	Remarks
1		Direction of rotation left, R=9.5, I=0 abs., make dialog selection, Transition to following element: R=2	
2		α1=–30 degrees	Observe angles in help screen!
3		CW direction of rotation, tangent to preced. R=2, J=4.65 abs.	
4		CCW direction of rotation, tangent to preced. R=3.2, I=11.5 abs., J=0 abs., make dialog selection, make dialog selection	
5		CW direction of rotation, tangent to preced. R=2, J=–4.65 abs., select dialog	
6		Tangent to previous element α1=–158 degrees, Y=–14.8 abs., α2=0 degrees	Observe angles in help screen!
7		All parameters, L=5, select dialog	
8		Y=5.7 abs.	
9		X=0 abs.	

6.6 Program simulation

6.6.1 Turning simulation

Function

The "Simulation" function operates in conjunction with turning technology.
With the "Simulation" function you can

- represent axis motions in graphic displays and
- trace the machining result on the screen as the workpiece is actually machined.

By activating Simulation, you can execute a contour on the screen in graphic form with or without machine axis motions (can be disabled by the PLC).

Display elements

The colors in the graphic display area signify the following:

- **Red** = Traversing path in the feed rate
- **Green** = Traversing path in rapid traverse
- **Yellow** = Cross-hair
  Polymarker (cutting edge),
  Workpiece symmetry axis

Cross-hair

Using the cross-hair, you can

- select the zoom center point and
- set the measuring points (for viewport).

Tool cutting edge

The position of the cutting edge corresponds to the definitions in the "Tool compensation" menu under softkey "Tool".
The tool path in the program block you are currently editing is simulated. The cutting edge is represented by a polymarker. The starting point of the polymarker corresponds to the starting point of the machine tool axes.

Coordinate system

The alignment of axes (coordinate system) is defined in machine data.

Machine manufacturer

Please see information supplied by machine tool manufacturer.
The display machine data are described in:

References

/FB1/K1: Mode Group, Program Operation, Chapter 4

Operating sequence

Simulation

Select a program, open it, and press the "Simulation" softkey.

The graphic simulation function is started when you press the "Cycle Start" key on the machine control panel.

The following softkey functions are provided:

Auto-zoom

This softkey automatically adjusts the display area to the displayed traversing motions.

To origin

You return to the initial display (size of the viewport when you select simulation). The viewport can be defined by the machine manufacturer in the machine data.

Display all

Optimizes the window for simulation purposes.

Zoom +

Zoom -

The current contents of the screen are displayed in a larger or smaller resolution when you press softkeys "ZOOM+" or "ZOOM-". Using the cursor keys, you can move the cross-hair to the selected center point of the window display.

Delete window

The current screen contents are deleted.

Cursor fine or Cursor coarse

You can alter the increments of the cursor key movements with the softkey "Cursor fine".

- Softkey is selected:

Cursor moves in "fine" increments.

- Softkey is not selected:

The cursor moves in "coarse" increments.

Close

Softkey "Close" ends the simulation.

Simulation is also aborted when you select a horizontal softkey.

6.6.2 Milling simulation before machining

Function

In automatic mode you can display your program graphically in the "Program test" function before machining, without traversing the machine axes.

Simulation graphic

The simulation graphic shows a representation of a workpiece being machined by a cylindrical tool. You can select different views via softkey, e.g.

- Top view
- Representation in three planes
- 3D representation (volume model)

Status displays

The status displays in the simulation graphic contain information

- about the actual axis coordinates and
- the block currently being processed.

Option

This function is an option and is only available with a color display. Milling simulation is only possible in the 1st channel.

Operating sequence

Prerequisite

- You select the program in automatic mode "Auto".
- In the Machine operating area under "Program control," the functions "Dry run feedrate" and "Program testing" (the machine is not moved while the program is run) are selected. If the "Dry run feedrate" function is active, the programmed feedrate is replaced by a defined dry run feedrate.
- Tool T0: Tool displayed in the graphics.
- Tool not identical with T0: An associated tool cutting edge must be selected.

Select a program of your choice and open it.

Press the "3D sim" softkey.

The program is started.
You can follow program execution on the screen.

6.6.3 Milling simulation during machining

Function

The current machining operation on the machine tool is simulated on the monitor of the control at the same time.

Option

This function is an option and is only available with a color display. Milling simulation is only possible in the 1st channel.

Operating sequence

Prerequisite

See previous section

Press the area switchover key and

3D sim.

the "3D sim." softkey.

Cycle Start

The program is started.
You can follow program execution on the screen.

You can start simulation at any time during the machining operation. Simulation is closed when you exit the graphic.

If you switch to another operating area, the current content of the graphic simulation is deleted.

Blank definition via input form

Details Settings

Select softkeys "Details" and "Settings" to open the blank definition window.
You can enter values for corner point 1 (front top left) and corner point 2 (back bottom left) of the blank (cube).

Alternative

With the softkey "Alternative" you can display/hide the view of the blank. If the view of the blank is disabled, the traversing paths are represented by broken-line graphics.

Define a blank via the NC program

As an alternative, you can define a blank in the NC program to be simulated.
Syntax:
`WRTPR("<String>")`
The following statements can be used in the "String":

- Rectangle: BLOCK(p1x, p1y,p1z,p2x,p2y,p2z)
  The positions correspond to the axis values of corner P1 (front top left) and P2 (back bottom right) of the rectangular blank.
  P1x = X value of corner P1
  p1y = Y value of corner P1
  p1z = Z value of corner P1

p2x = X value of corner P2
p2y = Y value of corner P2
p2z = Z value of corner P2

- Moving/rotating the graphic
  `FRAME(pv1,pv2,pv3,pd1,pd2,pd3)`
  pv1 = Moving the first axis
  pv2 = Moving the second axis
  pv3 = Moving the third axis
  pd1 = Rotating around the first axis
  pd2 = Rotating around the second axis
  pd3 = Rotating around the third axis
- Switching off the graphic: `END()`
- Restoring the unmachined blank: `CLEAN()`

Example

```
...
; DEFINITION of the blank
N100 WRTPR("BLOCK(0,0,0,80,100,-30)")
N110 ...
...
;DELETING the blank
N1000 WRTPR("CLEAN()")
...
```

References For additional procedures, please refer to the following documentation:
/BAS/ ShopMill Operator's Guide

6.7 Manage programs

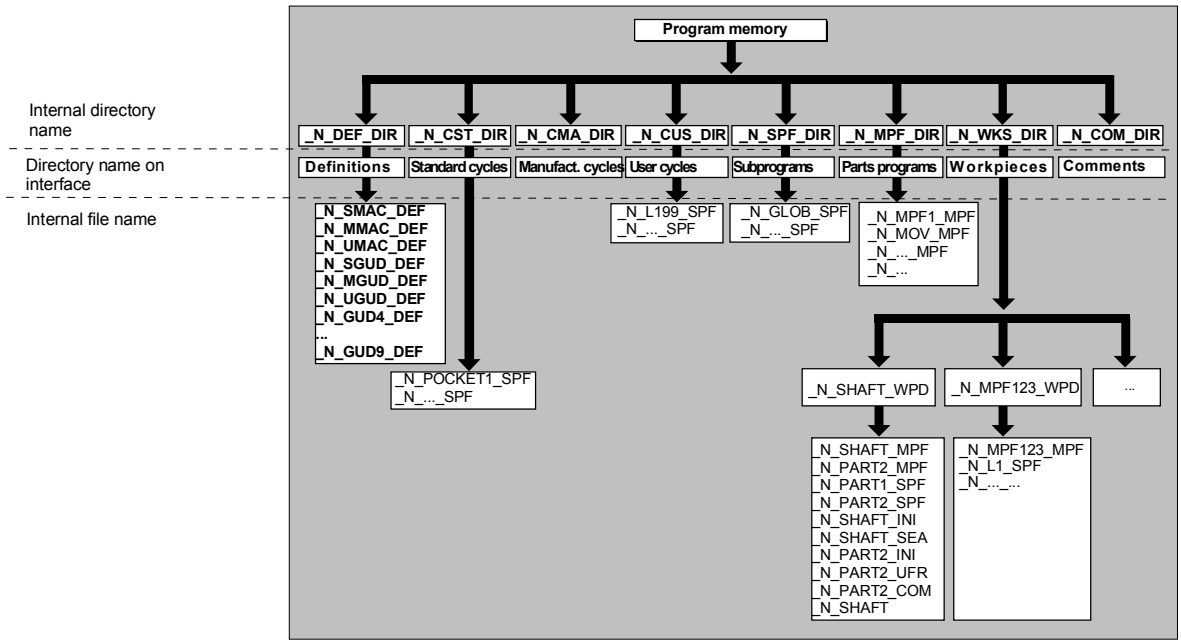
6.7.1 Overview

To allow you to handle data and programs flexibly, you can organize, store and display them according to different criteria.
The data/programs are stored in the NC memory by default. These programs are read in and out and executed via the RS-232-C interface.
In addition, further storage devices are available such as network drives, Compact Flash Cards and PCMCIA cards.

Program management Programs and files are stored in different directories:

- Subroutines
- Parts programs
- Workpieces
- Definitions
- Comments
- Standard cycles
- Manufacturer cycles
- User Cycles

The following diagram shows an example of directory contents:



6.7.2 File types, blocks and directories

File types

File types can be identified by the file name extension (e.g. .MPF).

name.MPF	Main program
name.SPF	Subroutine
name.TEA	Machine data
name.SEA	Setting data
name.TOA	Tool offsets
name.UFR	Zero offsets/frames
name.INI	Initialization files
name.COM	Comment
name.DEF	Definition of global user data and macros

Block

"Block" is the term given to any files required for creating and processing programs.

Program block

Program blocks contain the main and subprograms of the part programs.

Data block

NC data unit: Data blocks contain data definitions for global user data. These data can be initialized directly when they are defined.

Initialization block

Initialization blocks contain the default settings for data.
The initial block is an ".ini" file. It contains values for initializing, for example, machine, user and system data, etc.

Macro block

Macro blocks are used to program one or more instructions with a single new name. Macro definitions are stored in the following files in directory Definitions:

_N_SMAC_DEF	Siemens macro definitions
_N_MMAC_DEF	Machine manufacturer macro definitions
_N_UMAC_DEF	User macro definitions

Reserved definition names for macros

The following macro definitions can be stored as standard:

_N_SMAC_DEF	Macro definitions (Siemens)
_N_MMAC_DEF	Macro definitions (machine manufacturer)
_N_UMAC_DEF	Macro definitions (user)
_N_GUD1_DEF	Definitions for global data (Siemens)
_N_GUD2_DEF	Definitions for global data (machine manufacturer)
_N_GUD3_DEF	Definitions for global data (user)

Directory types

In addition to files, some directories may also have extensions:

name.DIR	General directory containing program and data and blocks, workpiece directories and other directories with identifier DIR.
name.WPD	Workpiece directories which contain program and data blocks that belong to a workpiece. (It must not contain another directory with the extension DIR or WPD.)
name.CLP	Clipboard directory: Files and directories of any type may be stored in here.

Workpiece directory

Workpiece directories (with extension .WPD) are set up in directory WCS.DIR.
A workpiece directory contains all files required for machining a workpiece.
These can be main programs, subprograms, any initialization programs and comment files.

Example:
Creation of a workpiece directory `SHAFT.WPD` that contains the following files:

`SHAFT.MPF`	Main program
`PART2.MPF`	Main program
`PART1.SPF`	Subroutine
`PART2.SPF`	Subroutine
`SHAFT.INI`	General initialization program of data for the workpiece
`SHAFT.SEA`	Setting data initialization program
`PART2.INI`	General initialization program of data for program part 2
`PART2.UFR`	Initialization program for frame data for program part 2
`SHAFT.COM`	Comment file

6.7.3 Handling files

Assigning values to data

A series of modules/data is permanently installed in the control during start-up.

References

The structure and handling of these files is described in the following documentation.
IAM/BE1: Expand the Operator Interface

Reading out data

You can save files via interfaces to external devices (programming device, diskette).

When saving files in punched tape/ASCII format, the entire path from which the file was saved is entered in the backup file.

The source path is specified in the second line:

File `SHAFT.MPF` was saved from the workpiece directory (`WCS.DIR`) under workpiece `SHAFT.WPD`.
Example:

```
%_N_SHAFT.MPF
;$PATH=/_N_WKS_DIR/_N_SHAFT_WPD
N10 G0 X… Z…
M2
```

The internal name of the file directories is e.g. _N_WCS_DIR.

For a complete backup of all data in a directory, the identifier COMPLETE is used for saving.
The complete backup of all data from all directories (INITIAL over all areas) is saved in the INI file _N_INITIAL_INI.

Import data

When you read in a file, the path entered when you saved the file is used. The system tries to read the file into the directory from which it was saved. If the path is missing, then files with file type SPF are stored in /SPF.DIR, files with extension .INI in the active working memory and all other files in /MPF.DIR. Files are immediately effective after import.

Activating data

Data can be activated/edited by loading files into the working memory. The exact time of activation depends on the type of data activated in the file.

References

/LIS/ Lists

For example, machine data can take effect (depending on type) either

1. Immediately or
2. on "RESET" or
3. on "Cycle-Start" or
4. on "Power ON" – when the control is switched on again.

Selecting a workpiece

A workpiece can be selected for machining in a channel.
see Section: "Select a Program on the NC for Execution", “Execute Program from Network Drive, Compact Flash Card, Disk”.
If a main program of the same name exists in the workpiece directory, it is automatically selected for execution. If you choose workpiece SHAFT.WPD, then the main program SHAFT.MPF is automatically selected.
If an .INI file of the same name exists, it is executed immediately (i.e. it is loaded into the working memory of the NC). Main programs with other names must be selected explicitly.

If a control has several channels, programs can be selected for processing and started from one part program for another channel.

Example:
The workpiece directory
/WCS.DIR/SHAFT.WPD
contains files
SHAFT.SPF and SHAFT.MPF.

Selection

When you select workpiece directory SHAFT.WPD you implicitly select program SHAFT.MPF.

Find path for program call

If the call path is not explicitly specified in the part program when a subroutine (or an initialization file) is called, the called program is located according to the predefined search strategy.
Case 1:
When a subprogram is called by
name **with specification of the file type** ("identifier" or "extension"), e.g. SHAFT.MPF,
the system searches through directories in the following order:

1.	Current directory / name.typ	Workpiece/standard directory MPF.DIR
2.	/SPF.DIR / name.typ	Global subprograms
3.	/CUS.DIR / name.typ	User cycles
4.	/CMA.DIR / name.typ	Manufacturer cycles
5.	/CST.DIR / name.typ	Standard cycles

Case 2:
When a subprogram is called by
name **without specifying the file type** ("identifier" or "extension"), e.g. SHAFT1,
the system searches through directories in the following order:

1.	Current directory / name	Workpiece/standard Directory MPF.DIR
2.	Current directory / name.SPF	
3.	Current directory / name.MPF	
4.	/SPF.DIR / name.SPF	Subroutines
5.	/CUS.DIR / name.SPF	User cycles
6.	/CMA.DIR / name.SPF	Manufacturer cycles
7.	/CST.DIR / name.SPF	Standard cycles

/PGA/ Programming Guide, Advanced

6.7.4 Create new workpiece/part program

Selecting a workpiece/part program

The following subsection describes how you can select workpieces and part programs in a directory. A selected file can then be called and edited in the text editor.

Operating sequence

Select workpiece/part program:

Workpieces
- Workpieces

Parts programs
- Parts programs

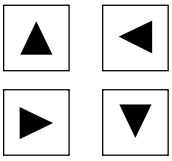
Sub-routines
- Subroutines

If "Logical drives" is configured, cycle softkeys are displayed after you press the ETC key.

Standard cycles
- Standard cycles

User cycles
- User cycles

Manufact. cycles
- Manufacturer cycles



Position the cursor in the directory on the desired file.
For each file, the file name, file type, length, date of creation/last change are displayed.
The file display properties can be set.
(See Chapter "Start-Up", "Settings" menu)

Call a part program:

Use the cursor to select a program in the program overview and press the "Input" key.
The text editor is displayed with the file you have selected.

You can now edit the part program.

Open a workpiece:

The workpiece directory is opened and the programs it contains displayed on the screen.

Create workpiece directory

You can set up various types of files such as main programs, initialization files, tool offsets, etc. in the new workpiece directory.

Operating sequence

Workpieces

The current overview of all workpiece directories appears on the screen.

New

Input window "New" is opened.
The cursor is located in the input field for the name of the new workpiece directory.

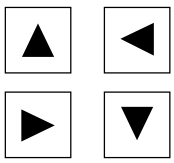
Enter the name of the new directory on the alphanumeric keyboard.
A new directory is set up in the workpiece overview.

Set up programs/data in a workpiece directory

This section explains how you can set up a new file for a part program or workpiece.

Operating sequence

The current overview of the workpiece directories stored on the NC appears.



Position the cursor on the required workpiece directory and open it.

An overview of the data and programs that have already been set up in the workpiece directory is displayed. If no data exists, an empty program overview appears.

New

A dialog box appears when you select softkey "New".

Enter the new file name. Program names may be a maximum of 24 characters in length. You can use any letters, digits or the underscore symbol (_). The file type is set by default.

OK

Then press the "OK" softkey.

The following file types can be entered:

File type	Meaning
.MPF	Main Program File
.SPF	Sub Program File
.TOA	Tool Offset Active
.INI	Initializing data
.COM	Comment file
.GUD	User data (global)
.TEA	NC-Machine data (Testing Data Active)
.SEA	Addresses with value assignment (Setting Data Active)
.LUD	User data (local)
.UFR	Zero point offset (User Frame)
.EEC	Spindle pitch/ sender error compensation.
.QEC	Quadrant error compensation
.CEC	Sag/angularity compensation

Parts programs or Sub-routines

Creating part programs in part program/subprogram directory:
You can set up main programs and subprograms by opening directories "Part programs" and "Subroutines".

New

Select softkey "New" to display a dialog window in which you can enter the names of the new main programs and subprograms.
The matching file type is automatically assigned in this case.

INPUT or OK

Then press the "OK" softkey or the "Input" key.

6.7.5 Execute a program on the NC for execution

Function

Workpieces and part programs must be selected for execution before you press the Cycle Start key.

Operating sequence

Select a program:

Parts programs

Use the cursor keys to select a program in the

program overview, e.g. part programs, and

then press softkey "Selection".

Program selection

The program name is displayed in the "Program name" window at the top right.

Select a workpiece:

A workpiece directory can be selected for machining in the currently active channel.

Workpieces

Use the cursor keys to select the workpiece

in the workpiece overview and then

Workpiece selection

press "Selection" softkey.

Once you have selected the workpiece, change back to the "Machine" "Automatic" operating area and select "Cycle start" to start machining. If you start machining in the "Program" operating area, you cannot track the cycle on the screen.

Machine manufacturer Please note information supplied by machine tool manufacturer!

References /IAM/IM2/ Installation & Start-Up Guide: HMI Embedded

6.7.6 Execute Program from network drive, compact flash card, disk

Function

If the "Network and disk drive management on PCU 20" option is enabled, you can connect the control to an external network drive connected to the PCU or another computer and select and run programs from there.
The function is also available in the “Services” operating area.

Precondition:

- The computer you want to connect is accessible/enabled.
- The connection is set up to the computer.
- The softkeys for linking computers have been configured via the machine data.

References /IAM/IM2, Installation & Start-Up Guide HMI Embedded

Operating sequence

Network drive | Compact flash card

When you select one of the configured softkeys, e.g. "Network drive", "Compact flash card" or "Disk", the Explorer appears on the screen with the files and programs of the external drive.

Disk

Select the program that you want to execute with the cursor and then press the "Execution from ext." softkey.

Execution from ext.

Further information

Direct execution from a network drive is not recommended, as the stability and dynamic response of all networks cannot be guaranteed.
Recommendation:
Copy the programs from the network onto the compact flash card on the PCU20 and run them from there.

6.7.7 Storing a program

Function

You can save the changes in a program that you have unloaded with the function "Save file".

Operating sequence

Save file

Changes are saved in the file loaded in the editor.

Further information

Please note that the changes to programs stored in the NC memory take immediate effect.

6.7.8 Enable workpiece/program

Function

The program overview indicates whether a workpiece or part program is enabled.
This means that It may be executed by the control (because it has already been debugged) when you select softkeys "Select program" and "Cycle Start".
When a new program is created, it can be enabled automatically. You can select this in the editor settings.

Operating sequence

To set the enable for a program or abort it, position the cursor on the desired workpiece or part program in the program overview.

Change enable

Select softkey "Change enable".
A cross indicating "Enable issued" appears behind the workpiece or part program.

(X) Enable issued (program is executable)

( ) No enabling (program must not be executed)

Further information

- The system checks whether a program may be executed when the program is called (after selection via operator input or from part program). If you want it to be enabled, you must enable it beforehand.

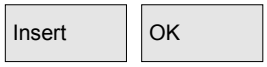
6.7.9 Copy file and insert

Function

This subsection explains how files can be copied.

Operating sequence

Position the cursor on the file that you want to copy and press the softkey "Copy".
The file is marked as the source for copying.



Press the softkey "Insert", enter a new name and confirm with "OK".

When you insert a file in a workpiece directory, you can change its file type with the "Select" key. The file types are automatically adjusted in the global part program and the global subroutine directory.

Further information

- Only files can be stored in a workpiece directory, not other workpiece directories.
- If the target specified is incorrect an error message is output.
- If a workpiece directory is copied, all the files that it contains are copied at the same time.

6.7.10 Rename file

Function

As regards files, you can alter their name as well as the associated file type.

Operating sequence

Position the cursor on the file you want to rename.

Rename

The "Rename" dialog window opens.

Enter the new name.

Only the workpiece is renamed, not the files it contains with the same name.

There are two ways of renaming files:

- Renaming the workpiece directory
- Renaming a file in the workpiece directory

Renaming a workpiece directory:
When you rename a workpiece directory, all the workpiece files under that directory that have the same name as the directory are renamed.
If a job list with the name of the directory exists, the instructions in that job list are also renamed.
Comment lines remain unchanged.

Example:
Workpiece directory `A.WPD` is renamed `B.WPD`:
All files with the name `A.XXX` are renamed to `B.XXX`, i.e. the extension is not altered.

Renaming a file in the workpiece directory:
If you rename the files in the workpiece directory, all files with the same name but a different extension are renamed.

6.7.11 Delete file

Function

This section explains how you can delete workpieces or files.

Operating sequence

Position the cursor on the workpiece or the file you want to delete.

Delete several files:
If you want to select several files, position the cursor on the first file, press the "Select" key and then place the cursor on the last file.
The files you have selected are highlighted.

The prompt "Do you really want to delete the file?" appears.

Confirm your input.

Further information

- You can only delete programs that are not currently running.
- If you want to delete a workpiece directory, make sure that none of the programs it contains is currently selected.
- If a workpiece directory is deleted, all the files that it contains are deleted at the same time.

6.7.12 "Workpiece template" function

If no _TEMPL_ exists for a workpiece, then you must create a workpiece named _TEMPL_. The files to be used as _TEMPL_ template files must be set up accordingly.

Operating sequence

If the workpiece already exists, it will be copied and renamed when you select "New". All files in this directory are also copied into the new directory.
TEST.MPF

Example:
Workpiece:
TEMPL
TEMPL.MPF
DATA.INI
TEST.MPF
Select softkey "New"
Name: Define "AXIS"
The new "Axis" workpiece is set up with the following files:
AXIS.MPF
DATA.INI

6.8 Memory info

Function

You can call a display showing the total available NC memory space.

Operating sequence

Memory info

When you press the "Memory info" softkey, the total free/used memory is displayed for
- NC memory
- Directories
- Files

in Mbytes.

6.9 EXTCALL

You can use the EXTCALL command to access files on network drives from a part program.
See:

References PG/ Programming Guide Advanced, Chapter 2

Supplementary conditions The following secondary conditions must be taken into account with EXTCALL calls:

- With EXTCALL you can only call files with the MPF or SPF extension from a network drive.
- The files and paths must adhere to the following NCK conventions: max. 25 characters for the name, 3 characters for the extension name.
- A program unit is found on a network drive with the EXTCALL command if
  - The search path points to the network drive with $SC_EXT_PROG_PATH - or to a directory on it. The program must be stored there directly; no other files must be located there. Any subdirectories are not searched.
    If you did not specify a drive, the call will be acknowledged negative.
  - The correct location of the program is specified in the EXTCALL call itself by means of a fully qualified path that can also point to a subdirectory of the network drive.
    If there is no file extension (MPF, SPF...), the HMI system will automatically add SPF.

Further information

Direct execution from a network drive is not recommended, as the stability and dynamic response of all networks cannot be guaranteed.

Recommendation:
Copy the programs from the network onto the compact flash card on the PCU20 and run them from there.

■

Services Operating Area

7.1 Function

Function

The "Services" Operating Area provides the following functions:

- Read data in/out
- Manage data
- Series start-up
- Error logging output

7.2 Directory structure

All files are organized in a directory structure.

Files in the NC memory are sorted according to keywords.

7.2.1 NC active data

NC-active data can be displayed and selected in the "Data" menu. They can be transferred, e.g. to a PC, via the RS-232 interface.

7.2.2 Directories

The following directories contain special files:

1. **Clipboard:**
   Files and directories of any type may be created/stored in the clipboard.
   They can also be selected for processing here.

Files must be transferred for storage on an external computer by means of a data transfer program such as PCIN.

2. **Workpiece:**
   All files (tool programs, tool data) needed to machine a workpiece can be stored in the "Workpieces" `(.WPD)` directory.
   Like a part program, a workpiece can be selected in the NC for machining.

When a workpiece is selected for machining, an INI file (if available) of the same name as the workpiece is loaded to the NC and the main part program with the same name as the workpiece is automatically selected.
If there is no part program/MPF with the same name, an error message is issued and the previously selected part program remains active.
If an INI block of the same name (e.g. for the activation of tool data), other initialization blocks can be executed.
Example:

SHAFT.WPD	is selected
SHAFT.MPF	is displayed as the selected program
SHAFT.INI	is loaded in the working memory of the NC and processed

7.2.3 Data selection

You can read the following selection of file types in or out via the RS-232 interface:

File type	Meaning
.MPF	Part Program (Main Program File)
.SPF	Subroutine (Sub Program File)
.TOA	Tool offset (Tool Offset Active)
.UFR	Zero point offset (User Frame)
.TEA	NC machine data (Testing Data Active)
.RPA	R parameter with value assignment (R Parameter Active)
.SEA	Addresses with value assignments (Setting Data Active)
.COM	Comment file
.INI	Initializing data
.GUD	User data (global)
.LUD	User data (local)
.WPD	Workpiece directory
.SYF	System files
.OPT	Options
.BOT	Booting files 611D
.DIR	Directory
.DEF	Definitions data
.CEC	Sag/angularity
.QEC	Quadrant error compensation
.EEC	Measuring system error compensation

Keywords/directories in the tree file structure

The different files are made available for transmission under the following headings:

- Data (general):
  - Option data
  - Machine data (all, NC-MD, channel MD, Axis MD)
  - Setting data
  - Tool offsets
  - Work offsets
  - Global user data
  - R parameters
- Start up data
  - NCK data
  - PLC data

- Display machine data
- Workpieces
- Parts programs
- Subroutines
- User cycles
- Manufacturer cycles
- Standard cycles
- Comments
- Definitions
- Feed drive
- Main Spindle Drive
- OEM

- System Data
  - ASUP1
  - ASUP2
  - IBN
  - OSTORE1
  - OSTORE2
  - Versions

- Log book
- Communications fault log
- Texts
- Machine configuration

If your control includes additional directories, these can be found in the file tree.

7.3 Formats for saving and importing data

Path name

The path name is automatically entered when files are saved (archived).
The path is named in the first line of a file:

```
;$PATH=/_N_WKS_DIR/_N_SHAFT_WPD
```

When the file is re-imported into the control, it is stored in this path.
If no path is specified, the files are read in to the currently active, selected directory or to the clipboard.

Example of file with path name:

```
%_N_SHAFT_MPF
;$PATH=/_N_WKS_DIR/_N_SHAFT_WPD
N10 G0 X... Z...
...
M2
```

Formats

Files can be saved to archive files in three different formats:

- Punched tape/ASCII format
- PC/binary format
- Punched tape/ISO format

7.3.1 Punched tape format

1. Only files with characters that can be displayed, i.e. files created in the text editor, can be saved. No binary data, however, can be saved.
2. Files in punched tape format can be edited with the text editor.
3. Files can be set up externally in punched tape format provided that they are formatted in compliance with the format specified below.
4. If a file is set up manually, it must begin with %<name>. "%" must be typed in the first column of the first line. An archive in punched tape format may contain several files, each of which must begin with `%<name>`.

The structure of archive files in punched tape format is as follows:

```
<leader>          ;can be included
%1st file name
;$PATH=1st path name ;can be included
1st block         NL          ;contents of file 1
2nd block         NL
...               NL
last block        NL

%2nd file name
;$PATH=2nd path name;can be included
1st block         NL          ;contents of file 2
...               NL
last block        NL
...                           ;contents of file n
last block        NL
<trailer>         ;can be included
```

<leader>

Information of any type (characters with ANSI values < ANSI value 32 (blank)) which is not part of the useful data on the punched tape. They might be positioned at the beginning of the tape so that it can be inserted into the punched tape reader.

When the archive file is read, a check is made to determine whether it was saved with a leader. If it was, then it is read in again with a leader.

NL

Character for block end/new line; ANSI value 10 (0x0A)

%

Identifier positioned in front of a file name.
The identifier must be positioned in the first column of the relevant line (at beginning of block).

File names

1. File names can contain the characters `0...9, A...Z, a...z` or _ and must not exceed 24 characters in total.
2. File names must have a 3-character extension (_xxx).
3. Data in punch tape format can be generated externally or processed with an editor. The file name of a file stored internally in the NC memory starts with `"_N_"`.
   A file in punched tape format begins with **%**<name>, "%" must appear in the first column of the first line.

Examples:	%_N_SHAFT123_MPF = parts program SHAFT123 **or** %flange3_MPF = parts program flange3
;$PATH=	Path statement; Identifier in front of a path name. The path statement must always be programmed as the next block after the file name. The ";" character in the path statement must be positioned in the first column of the relevant line (at beginning of block).
Path names	1. Path names end with `_DIR` (directory) or `_WPD` (workpiece). 2. Path names can contain the characters `0...9, A...Z, a...z` or `_`. 3. Paths must be specified in full (beginning with "`/`"). The separator for the directory hierarchy is "`/`". 4. A path name in punched tape format begins with `;$PATH=`<path name> in the first column of the program. Path names in punched tape format start with `_N_` and end in `_DIR` (any directory) or `_WPD` (workpieces directory).
Example:	;$PATH=/_N_WCS_DIR/_N_PIVOT_WPD Workpiece directory PIVOT in directory Workpieces
	The data listed after the file name/path name belong to the file with the name specified after "`%`" in the directory specified after "`;$PATH=`".
<trailer>	Any information (characters with ANSI values < ANSI value 32 (blank) and not equal to ANSI value 10 (0x0A)) which is not part of the useful tape data.
Search strategy when no path is indicated	If no path is entered in the punched tape format, the specified file name must be interpreted when the file is read into the control so that the file can be stored in a suitable position in the file tree.

Files are stored in the file tree according to the following strategy:

File name in tape format	Converted internal file name	Interpreted internal path	Stored in directory
%*_INI	_N_*_INI	/_N_NC_ACT_DIR	NC active data
%_N_*_XXX	_N_*_XXX	/_N_XXX_DIR /_N_NC_ACT_DIR	XXX /_N_NC_DIR
%MPFn	_N_MPFn_MPF	/_N_MPF_DIR	Parts programs
%SPFn	_N_SPFn_SPF	/_N_SPF_DIR	Subroutines
%Ln	_N_SPFn_MPF	/_N_SPF_DIR	Subroutines
%*	_N_*_MPF	/_N_CLIP_DIR	Clipboard

* = any file name
n = any program number (e.g. MPF123)

- The search strategy is applied only if no path has been named. Paths detected using the search strategy are otherwise overwritten by the `";$PATH="` statement.
- Spaces in the name are ignored.

Examples

1. *.MPF files
   - PC format:
     Part program Directory: Part program
     `%MPF123` (`/_N_MPF_DIR`)
   - Punched tape format:
     Part program Directory: Part program
     `%_N_MPF_MPF ;$PATH=/_N_MPF_DIR`

2. *.INI files
   - PC format:
     Part program Directory: NC active data
     `%COMPLETE_TEA_INI` (`/_N_NC_ACT_DIR`)
   - Punched tape format:
     Part program Directory: NC-active data
     `%_N_COMPLETE_TEA_INI ;$PATH=/_N_NC_ACT_DIR`

3. Part programs with names that cannot be assigned
   - PC format:
     Part program Directory: Clipboard
     `%HUGO` (`/_N_CLIP_DIR`)
   - Punched tape format:
     Part program Directory: Clipboard
     `%_N_HUGO_MPF ;$PATH=/_N_CLIP_DIR`

7.3.2 PC format binary format

Files which contain non-displayable characters/binary format can only be saved in PC format.

References /IAD/Start-up: Section "Line checksum".

- If you save files in PC format and then edit them with a text editor, you will not be able to re-import them again. The file cannot be edited because the checksum will then no longer be correct.
- Start-up and update data must always be saved in PC format.

7.4 RS-232 C interface parameters

Protocol The following protocols are supported for transmission via RS-232-C:

- XON/XOFF and RTS/CTS,
- Software Flow Control and Hardware Flow Control

XON/XOFF

It is possible to set the two modes on the user interface for RS-232-C transmissions, i.e. Wait for Xon for data receive and Send Xon for data transmission. The default setting is H11 or H13.

Input: Is done by selecting in "Interface" display with the cursor keys and the "Input" key.

One possible way of controlling transfer is to use control characters XON (DC1, DEVICE CONTROL 1) and XOFF (DC3). If the buffer of the peripheral device is full, it sends XOFF and XON as soon as it can receive data again (= default).

RTS/CTS

The signal RTS (Request To Send) controls the transmission of the data transmission equipment.

Active: Data can be transmitted.

Passive: The signal CTS (Clear to send) is an acknowledgement signal for RTS that the Data transmission equipment is ready to transmit.

Baud rate

Input: By selecting in the "Interface" display under "baud rate" with the selection key

300 Baud
600 Baud
1200 Baud
2400 Baud
4800 Baud
9600 Baud (default)
19200 Baud
:

Data bits

Number of data bits for asynchronous transmission.
Input: By selecting in the "Interface" display under "Data bits"

- 7 data bits
- 8 data bits (default)

Parity

Parity bits are used to detect errors:
The parity bits are added to the coded characters to make the number of places set to "1" an odd number (odd parity) or an even number (even parity).
Input: Selected in the "Interface" display under "Parity"

- No parity (= default)
- Even parity
- Odd parity

Stop bits

Number of stop bits for asynchronous transmission.
Input: Selected in the "Interface" display under "Stop bits"

- 1 Stop bit (= default)
- 2 Stop bits

Special functions

The following special functions are also provided. These can be activated in the "Interface" display.

A checkbox with a cross in it means: special function active.

Overwrite with confirmation only

☒	Active:	On reading in, a check is made to determine whether the file in the NC already exists.
☐	Inactive:	Existing files are overwritten without query.

Block end with CR LF

☒	Active:	During output in punched tape format A CR character (carriage return, hexadecimal 0D) is inserted after every LF (Linefeed).
☐	Inactive:	No CR inserted.

Stop with end of transmission character

☒	Active:	Text mode: The end of transmission character is active.
☐	Inactive:	Binary mode The end of transmission is not evaluated. The standard value for the transmission end character is hexadecimal 1A.

Evaluate DSR signal

☒	Active:	Transmission is interrupted if DSR signal (connection 6 on X6) is missing.
☐	Inactive:	DSR signal has no effect.

Leader and trailer

☒	Active:	Leader ignored during input. During output 120x0(Hex) is displayed. (Feed before and after the data)
☐	Inactive:	Both lead and trailer are read in. No leader 0(hex) on output. Read in is automatically recognized for all MMCs.

Cycle watchdog

☒	Active:	During transmission problems or end of transmission (without transmission end character) the transmission is terminated after 5 seconds. The time monitoring function is controlled by a time emitter that is started with the first character and reset with every transmitted character.
☐	Inactive:	Transmission is not aborted.

Punch tape format

Instead of binary format you can choose punched tape or punched tape/ISO format, i.e. programs are output in accordance with DIN 66025

e.g. SINUMERIK 840D programs: Start activated by % <file name>, %**MPF**<xxx> or %**SPF**<xxx>.

7.4.1 Interface parameterization

Parameters for serial printer

Default: RS-232-C printer
A printer with a serial interface is connected with a suitable cable (cable check to CTS).

Interface	COM 1		Selection of interface: COM 1 or COM 2
Device type	RTS-CTS		Start with XON
Baud rate	9600		Program start with LF
Stop bits	1	X	Block end with CR LF
Parity	None	X	Stop with end of transmission end character
Data bits	8		Evaluate DSR signal
XON	11 (H)		Leader and trailer
XOFF	13 (H)	X	Punch tape format
Transmission end	0C (FormFeed)		Cycle watchdog

Parameters for archiving with PG/PC

Default: RS-232-C PG/PC

Interface	COM 1		Selection of interface: COM 1 or COM 2
Device type	RTS-CTS		Start with XON
Baud rate	9600		Program start with LF
Stop bits	1		Block end with CR LF
Parity	None		Stop with end of transmission end character
Data bits	8	X	Evaluate DSR signal
XON	11 (H)		Leader and trailer
XOFF	13 (H)		Punch tape format
Transmission end	1A	X	Cycle watchdog

This setting allows files in SINUMERIK 840D PC format to be archived and imported.
"Stop with end of transmission character" must not be selected for transmission of MSD and FDD files.
Other settings are possible with ASCII data. They must match the settings on the programming device. Cable 6FX 2002-1AA01- is used for this purpose.

Parameters for DIN programs

Default: RS-232-C user

Interface	COM 1		Selection of interface: COM 1 or COM 2
Device type	RTS-CTS		Start with XON
Baud rate	9600		Program start with LF
Stop bits	1	X	Block end with CR LF
Parity None	X		Stop with end of transmission end character
Data bits	8	X	Evaluate DSR signal
XON	11		Leader and trailer
XOFF	13	X	Punch tape format
Transmission end	1A		Cycle watchdog

With this setting, programs are read in conforming to DIN (beginning with %).

Punched tape input/ output

The "with leader and trailer" checkbox must be activated for tape readers and punches.
If the punched tape reader is controlled by CTS, the "Stop with end of transmission character" checkbox must also be activated.

Device type	RTS-CTS		Start with XON
Baud rate	9600		Program start with LF
Stop bits	2		Block end with CR LF
Parity None	X		Stop with end of transmission end character
Data bits	8	X	Evaluate DSR signal
XON 00			Leader and trailer
XOFF 00	X		Punch tape format
Transmission end	00	X	Cycle watchdog

References

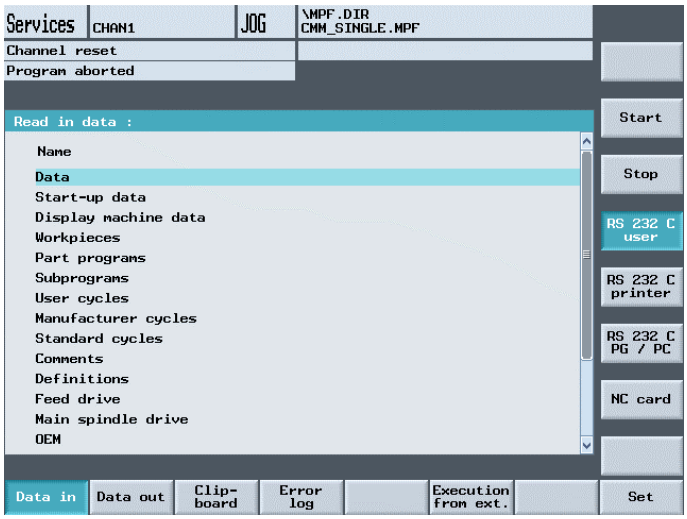
/FB1/K4/ Description of Functions, Basic Machine: Communication

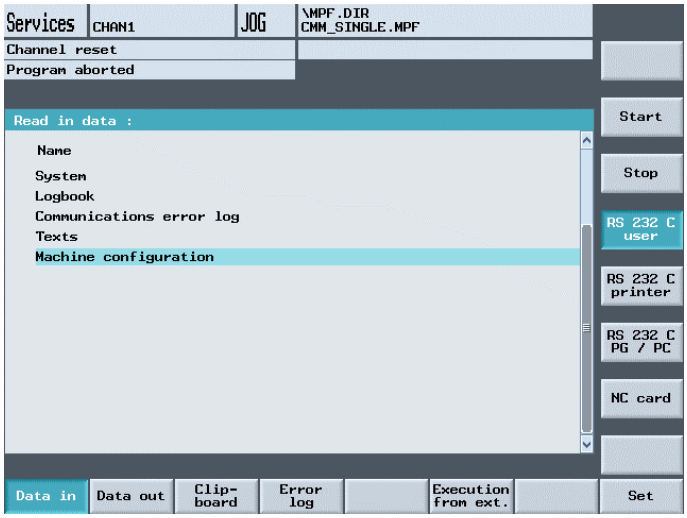
Reading in binary data (FDD, MSD)

Device type	RTS-CTS		Start with XON
Baud rate	9600		Program start with LF
Stop bits	1		Block end with CR LF
Parity None			Stop with end of transmission end character
Data bits	8		Evaluate DSR signal
XON			Leader and trailer
XOFF			Punch tape format
Transmission end	00		Cycle watchdog

7.5 Main services screen

All transferable data/programs are displayed in the "Services" basic display.





Horizontal softkeys

Data in	The "Read in data" menu is opened.
Data out	The "Read out data" menu is opened.
Clipboard	The "Clipboard" menu is opened.
Error log	You obtain information about the data transmission which has taken place.
Execution from ext.	Step-by-step loading and execution of external programs can be initiated in this screen.

Set

The interface parameters of the currently selected interface can be changed.

Vertical softkeys

Start

Data read-out is started.

Stop

The data output operation is aborted.

You can select the RS-232 interface via

RS 232 C user

- RS-232-C user

RS 232 C printer

- RS-232-C printer

RS 232 C PG/PC

- RS-232-C PG/PC
- is selected.

NC card

Data can be created, imported or deleted on the NC card.

additional

If the "logical drives" option is set up, the softkeys are displayed (max of 4 drive connections) via the softkey "additional".

The following applies in the "Services" operating area:

- Changes resulting from data transmissions are accepted and stored. Any modifications are thus retained when the NC powers up again.
- The selected window is closed. The next data tree display is displayed in the selected view.

7.5.1 Setting the interface

Function

You can output files to an external data storage device or read them in from there via the RS-232 interface. The RS-232 interface and your data backup device must be compatible. The control provides you with an input screen form for this purpose in which you can define the specific data for your device.

You have the option of selecting three different RS-232 interface parameter sets:

1. RS-232-C user
2. RS-232-C printer
3. RS-232-C PG/PC

Operating sequence

Select one of the three softkeys for the RS-232 interface. The softkey remains marked so that you can see what you have selected.

RS 232 C user

- RS-232-C user

RS 232 C printer

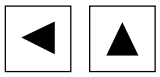
- RS-232-C printer

RS 232 C PG/PC

- RS-232-C PG/PC

Set

Press softkey "Set" if you wish to alter the interface parameters for the currently selected interface (softkey is highlighted). The name of the interface to be set up is displayed in the title.



Position the cursor on the input fields and enter the required values.

The values of parameters "Device type", "Baud rate", "Stop bits", "Parity" and "Data bits" are selected with the "Select key".

You can also activate and deactivate special functions with the "Select key".

Save settings

Close the parameter setting display by pressing the "Save settings" softkey.

The data for the interface are stored with the values you have entered.
(See also Section "Interface parameters")

7.5.2 Reading in data via RS-232 interface

Function

It is possible to read data into a particular directory or into the clipboard via the RS-232 interface.

Operating sequence

The interface for your data unit is set correctly.

Data in

When you press softkey "Data in", the currently selected RS-232-C interface parameter settings are stored.

Via the vertical softkey bar select the interface:

RS 232 C user

- RS-232-C user

RS 232 C PG/PC

- RS-232-C PG/PC

◀ ▲ ▼ ▶

Position the cursor on the desired directory in the directory tree.

Save settings

You return to the overview of the higher level directory by pressing the "Save settings" softkey.

⋀

You can discard your settings by pressing the Recall key.

Start

When you press the "Start" softkey, data are read in and stored in the specified path.

☒ Path/workpiece from archive file

To archive files using the "Data out" function, the directory paths from which the files were read are stored with the other data in archiving format.
The stored paths are interpreted on import and the files are copied back to the directories from which they were backed up (NC default `MPF.DIR`).

In punched tape format, the `;$path` = statement is applied.

☐ Path/workpiece from archive file

Irrespective of the archived path name, all files are stored in the directory selected beforehand with the "Direction keys".

☒ Readin to the clipboard

All archived data are stored in the clipboard regardless of the archive path name.

OK

The data are read in. The message "Transmission in progress" is displayed.
The display shows "Path/workpiece", "File" and the number of transmitted "Bytes".

Stop

You can interrupt data import at any time by pressing the "Stop" key.

Start

If you press softkey "Start" again, the data import recommences from the beginning.

Further information

- The control system cannot call specific data for import.
- If you have selected the "Overwrite and confirm" option, existing files are overwritten after your confirmation. On rejection, reading in continues with the next file.
- Only data that have an extension known to the system can be read in.
- When importing with the setting "Without file end character" or "Without timeout," transmission must be stopped with the STOP softkey.
- Transmission is not complete until the window "RS-232-C running" is closed.
- If a program is overwritten when data are read in, the NC deletes the program at the beginning of the transmission operation and inserts the program again when all data have been transferred.

Import of machine data

The BOOT files/`INITIAL.INI` initialization file produce the basic setting of the machine.

- The "Path from workpiece/archive" box must be checked for the RS-232 interface before the Start command is given.
  This applies both to data in archive format and in punched tape format.
- An "NC Reset" must then be carried out to make the machine data operative.

7.5.3 Inserting data from the clipboard

Function

You can store data from the clipboard in a new directory, copy or delete them.

Operating sequence

Data have been read into the "Clipboard" directory via the "Data in" interface.

Clipboard

Select softkey "Clipboard".
The vertical softkey bar changes.
The cursor is positioned on a file in the "Clipboard" window. The selected window is active.

Select source:
Position the cursor on the file that you wish to place in the data structure of the control.

Position the cursor in the top window by selecting the "Window selection" key. The top window is then active.

Select target:

Position the cursor on the target directory in which you wish to place the file you have just located.

Copy and paste

A confirmation window is displayed.

OK

If you wish to retain the file name stored in the clipboard, confirm by pressing softkey "OK".

If you want to use another file name, enter the new name and terminate your input by pressing softkey "OK". The file is copied into the target directory and is not deleted from the clipboard.

Back

Go back to the directory overview by pressing softkey "Back".

Deletion

Delete

You can only delete data from the clipboard with the "Delete" key.

Files stored in the clipboard are not automatically deleted when placed in the data structure.
You must ensure that the clipboard is cleared so that it does not take up too much memory space unnecessarily.

7.5.4 Reading out data via RS-232 interface

Function

You can read data out of the control system via the RS-232 interface to a device (e.g. PC) connected to the interface.

Operating sequence

The interface for your data unit is set correctly.

Data out

The "Data out" window is opened.

Via the vertical softkey bar select the interface:

RS 232 C user
- RS-232-C user

RS 232 C PG/PC
- RS-232-C PG/PC

RS 232 C printer
- RS-232-C printer

Position the cursor on the desired directory in the directory tree.

Save settings

You return to the overview of the higher level directory by pressing the "Save settings" softkey.

You can discard your settings by pressing the Recall key.

Start

The data are read out. The message "Transmission in progress" is displayed.
The display shows "Path/workpiece", "File" and the number of transmitted "Bytes".

Stop

You can interrupt data export at any time by pressing the "Stop" key.

Start

If you press softkey "Start" again, the data export recommences from the beginning.

7.5.5 Reading out PLC alarm texts and cycle texts

Function

PLC alarm texts, cycle texts and cycle alarm texts can be read out.

Operating sequence

Data out

The "Data out" window is opened.

A screen appears where you can select your desired language.
You can choose any of the following texts from "Texts":

- PLC alarm texts (user)
- PLC alarm texts (standard)
- Standard cycles
- User cycles
- Manufacturer cycles
- Cycle alarm texts (user)
- Cycle alarm texts (standard)

7.5.6 Outputting error/transmission log

Function

A log listing data that have been imported and exported can be output in the Services operating area.
The log contains

1. for files to be output
   - the file name complete with pathname, and
   - an error acknowledgment.

2. for files to be input
   - the file names and the 1st line, which generally contains the path name `;$PATH=...`, and
   - an error acknowledgment.

Transmission messages The following messages may appear during transmission:

"OK" Transmission has been terminated correctly.

"ERR EOF"	The end-of-text character has been received but the archive file is not complete.
"Time Out"	Transmission terminated by time-out. Archive: Not completely transmitted, last file not stored. Punched tape: Completeness cannot be checked, last file stored.
"User Abort"	Transmission terminated with "Stop" key. Archive: Not completely transmitted, last file not stored. Punched tape: Completeness cannot be checked, last file stored.
"Error Com 1"	Error at COM1 port overrun buffer: Overflow at input buffer overrun: Overflow at COM1 port parity error: Parity error or frame error: Frame error (data/stop bits/data transfer rate) broken line/no DSR: DSR signal missing (wire break) or BREAK (interruption) was received.
"NC/PLC Err or xxyyzzzz"	Error message from NC: xxyy Error code and error class signaled by the NC zzzz HMI Embedded internal error number The NC cause of error is logged together with a short single-line text.
"Error DATA"	Data errors: 1. Files with leader/without leader read in, or 2. Files sent in punched tape format without file names (PCIN).
"Error File Name"	The file name or path does not follow the naming conventions of the NC, e.g. special characters in the name or no 3-character extension.
"Tape format illegal"	Drive data (binary-coded data) can only be saved in 840D archive format (punched tape format not active).
"Tape format required"	Output of logs in punched tape format only.
"Rem CREG"	Reset Register X39: Note that the RS-232 was reinitialized.

Operating sequence

Error log

You can display information about the data transfer operation by pressing softkey "Error log".

7.5.7 Execution from external source via RS-232 interface

Function

You can download external programs via the RS-232 interface to the NC and edit them immediately. The buffer size for the part program blocks temporarily stored in the NC depends on the NC memory and its distribution.

- `H=...` contains, as well as
- an error acknowledgment.

Operating sequence

All downloadable data/programs are displayed.

▼

Select a range, e.g. part program and

Execution from ext.

Press softkey "Execution from ext.".

Start

Execution commences after "Start".

7.5.8 Reading ISO programs in/out via RS-232 C

Function

ISO programs can be read in and out in punched tape format.

Further information

FANUC 0 control system programs can be read in and out as ISO programs.

The punched tape format for ISO programs (ISO punched tape format) is not the same as Siemens HMI Embedded punched tape format.

The first line of a punched tape in ISO format must have the following format: %<Title>LF or %<Title>CRLF.
The title can be omitted and blanks can be skipped. The title may not start with one of the following characters: 0...9, a...z, A...Z or _.
No title is generated when a punched tape is generated in ISO format.

Siemens program headers are introduced by %<Name> and pathname ;PATH=<Path> in the next block.
ISO program headers are characterized by O<xxxx (Title)> O or : :<xxxx (Title)> without pathname in the next block.
x stands for a number between 0 and 9. Between one and four digits can be specified, leading zeros can be omitted.
During export, ISO program headers are only tagged with O<...> and not with :<...>.

Retrieving

Data in

The procedure for importing a punched tape in ISO format is the same as the procedure used to import a regular punched tape archive in the "Services" operating area with "Data in". During the import, the system detects automatically whether the archive to be imported is stored in binary/PC, punched tape or ISO punched tape format.

Imported ISO programs are stored as main programs on the NC. You must set the read-in directory every time you import programs to the "Services" area by selecting → "Data In" → "Start". If "Path from workpiece/archive" is selected, ISO programs are stored in the selected directory (e.g. workpiece xxx) or in the default NC directory (MPF.DIR); DIN programs are stored in accordance with their specified path.

ISO punched tape with two ISO programs:

```
%
01026(HYDRAULIC BLOCK)
N20 G00 G80 G90 G40 G17
N40(NC-SPOT DRILL)T01 M06
N50 G55 G43 Z20. H01 S1000 F100 M03
N55 X10. Y-8. M08 T02
(...)
N690 Y-43.
N700 G80 Z35.
N710 T00 M66
N715 G53 Y0. Z0.
N720 M30
:1127(ANGLE)
N10(2. SPEED RANGE)
N20 G00 G80 G90 G40 G17
N120(SPI-BO 11)T01 M06
N130 G55 G43 Z20. H01 S2300 F460 M03
(...)
N180 Y-72.
N190 G80 Z35.
N195 T00 M66
N200 G53 Y0. Z0.
N210 M30
%
```

This punched tape generates two programs when imported: `_N_1026_MPF` and `_N_1127_MPF`; the title is retained after the program number:

Program _N_1026_MPF:

```
(HYDRAULIC BLOCK)
N20 G00 G80 G90 G40 G17
N40(NC-SPOT DRILL)T01 M06
(...)
N710 T00 M66
N715 G53 Y0. Z0.
N720 M30
```

Program _N_1127_MPF:

```
(ANGLE)
N10(2nd SPEED RANGE)
N20 G00 G80 G90 G40 G17
(...)
N200 G53 Y0. Z0.
N210 M30
```

Export

Data out

The procedure for generating a punched tape in ISO format is the same as the procedure used to generate a regular punched tape archive in the "Services" operating area with "Data out". The current output format determines whether the archive is created in binary/PC, punched tape or ISO punched tape format.
The output format can be modified in the "Services" operating area with "Set".
A Select field offers the setting options
Punched Tape Format, Punched Tape Format/ISO or Binary Format (PC format).

If both ISO programs and Siemens programs are selected for the creation of an ISO punched tape archive, an ISO punched tape is generated without an alarm or message output; the punched tape contains Siemens program headers in addition to the ISO program headers.
If a Siemens program is followed by an ISO program, an %<LF> or %<CR><LF> is inserted in front of the ISO program header, depending on the output format, because the character string O<four digits> or :<four digits> cannot uniquely be assigned to a new program in DIN-Code.
These "mixed" ISO punched tape archives can be read back in by HMI Embedded; however, reading the archives in to other types of control will result in premature termination due to the % character (% character in ISO format indicates tape end).

```
%
%_N_TEST1_MPF
;$PATH=/_N_WKS_DIR/_N_TEST_WPD
N40 G01 X150 Y150 Z150 F6000
N50 G90 G0 X0 Y0 Z0 G53
; ...
N500 G02 z100 x50 k-50 i0
N510 z50 x100 k0 i50
M30     ;Transfer from Siemens prog. to Siemens prog.
%_N_TEST2_MPF
;$PATH=/_N_WKS_DIR/_N_TEST_WPD
N40 G01 X150 Y150 Z150 F6000
```

```
; ...
M30     ;Transfer from Siemens prog. to ISO prog.
%
O1127(ANGLE)
N10(2nd SPEED RANGE)
N20 G00 G80 G90 G40 G17
(...)
N200 G53 Y0. Z0.
N210 M30
%
```

If both ISO programs and Siemens programs are selected for the creation of a Siemens punched tape archive, a conventional punched tape is generated which contains only Siemens program headers, i.e. the ISO programs contain Siemens program headers.

```
%_N_TEST1_MPF
;$PATH=/_N_WKS_DIR/_N_TEST_WPD
N40 G01 X150 Y150 Z150 F6000
N50 G90 G0 X0 Y0 Z0 G53
; ...
N500 G02 z100 x50 k-50 i0
N510 z50 x100 k0 i50
M30     ;Transfer from Siemens prog. to Siemens prog.
%_N_TEST2_MPF
;$PATH=/_N_WKS_DIR/_N_TEST_WPD
N40 G01 X150 Y150 Z150 F6000
; ...
M30     ;Transfer from Siemens prog. to ISO prog.
%_N_1127_MPF
;$PATH=/_N_WKS_DIR/_N_TEST_WPD
(ANGLE)
N10(2nd SPEED RANGE)
N20 G00 G80 G90 G40 G17
(...)
N200 G53 Y0. Z0.
N210 M30
```

The difference is irrelevant for archives in binary format.

Further information

Binary files cannot be output in ISO punched tape format.
The display indicators differ in the use of ISO, particularly with regard to the representation of H numbers.

7.5.9 Execute program from network drive, Compact Flash Card, disk

Function

If the "Network and disk drive management on PCU 20" option is enabled, you can connect the control to an external network drive (up to a max. of 4 drives) connected to the PCU or another computer and select and run programs from there.
The function is also available in the "Program" operating area.

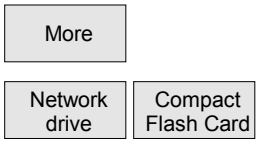
Prerequisite:

- The computer you want to connect is accessible/enabled.
- The connection is set up to the computer.
- The softkeys for linking computers have been configured via the machine data.

References /IAM/IM2, Installation & Start-Up Guide HMI Embedded

Operating sequence

Use the "More" softkey to display the configured softkeys for the external drive or computer on a second vertical bar.



When you select one of the configured softkeys, e.g. "Network drive", "Compact Flash Card" or "Disk", the Explorer appears on the screen with the files and programs of the external drive.

Select the program that you want to execute with the cursor and then press the "Start" softkey.

Further information

Direct execution from a network drive is not recommended, as the stability and dynamic response of all networks cannot be guaranteed.
Recommendation:
Copy the programs from the network onto the compact flash card on the PCU20 and run them from there.

7.5.10 Restoring the original status via NC card

Function

The free memory on the NC card (PCMCIA card) can be used to store a start-up archive.
The archive can be loaded onto the NC card with SINUCOPY-FFS (on an external PG/PC).

The series machine start-up archive can be stored directly with the name "Original" in HMI Embedded on the NC card (see Series machine start-up - creating a file).

Operating sequence

Precondition:
The start-up archive with the name _N_ORIGINAL_ARC is located on the NC card (in the directory _N_NC_CARD_DIR_N_ARC_DIR).

Various archives can be selected and generated

Select the "ETC key" in the main Services screen and then the "Original state" softkey.

When you press the softkey, the protocol window appears with the query: "Series start-up archive: Perform series start-up?". After confirmation, the data are imported.

Caution
All user-specific NC data (and PLC data depending on contents) will be deleted and replaced by the data from the archive.

Further information

Users with access authorization to level 3 and higher can access this softkey. It is displayed only if archive _N_ORIGINAL_ARC is stored on the NC card.

7.5.11 Saving data to NC card

Function

For saving data to NC-Card there is at least 3MB (or 8MB Memory Flash Card) available.
The files listed in the data overview ("Organize data" window) can be saved to the NC card.
Cycle alarm texts can be stored as well as start-up data, workpieces, etc.
Precondition: Directory "ARC.DIR" must already exist on the card.

Operating sequence

Data out

When you select the "Data out" softkey, the directories or files that you can select by paging and the cursor key are displayed; activate "Organize data" window if appropriate.

PAGE DOWN ▼

INPUT

You can open the directory or file with the "Input" key.

NC card

Select softkey "NC card"

Start

When you select softkey "Start" the "Create archive" window appears. Enter a name and confirm with "OK".

OK

If you want to read these files back from the NC card, follow the same sequence in reverse order.

7.5.12 Series start-up

Function

You can archive the current software on your machine and import it to other machines with series start-up.
You can choose which data you want to archive:

- NCK data with/without compensation data
- Loadable compile cycles
- PLC data

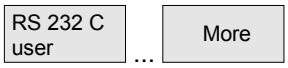
If you want to import data to other machines, you should not archive the compensation data along with other data, because compensation data is machine specific.
The drive data (contained in the NCK data) are saved in binary format; i.e. you cannot change the drive data.

You can archive data via the RS-232 interface or archive it to the NC Card or to other drives connected with the PCU (logical drives).
If you want to archive data via the RS-232 interface, you must set the "binary" format.

Operating sequence

Press the "ETC" key and then the "Series start-up" softkey.

You can choose which data you want to archive.



You can choose where you want to save the archive.

Press "Start" softkey.

Enter the archive name.

The selected data are saved in an archive file.

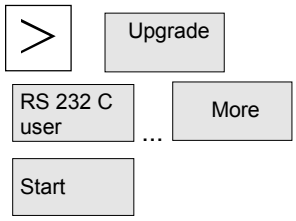
The archive file can be read in via the "Data in" function.

7.5.13 Upgrading

Function

If you want to upgrade your machine, you can archive the NCK data beforehand. In this way, the drive data is saved in ASCII format, i.e. you can change the drive data.
You can archive data via the RS-232 interface or archive it to the NC Card or to other drives connected with the PCU (logical drives).

Operating sequence



Press the "ETC" key and the "Upgrade" softkey.

Choose where you want to save the archive.

Press the "Start" softkey, and enter the archive name.

The NCK data (including drive data) are saved in an archive file.

The archive file can be read in via the "Data in" function.

7.5.14 Manage programs from network and disk drives

Function

If the "Network and disk drive management on PCU 20" option is enabled, you can connect the control to an external network drive connected to the PCU or another computer and select and run programs from there.
This function is also available in the "Program" operating area.

Operating sequence

See Chapter 6: Manage Programs from Network and Disk Drives

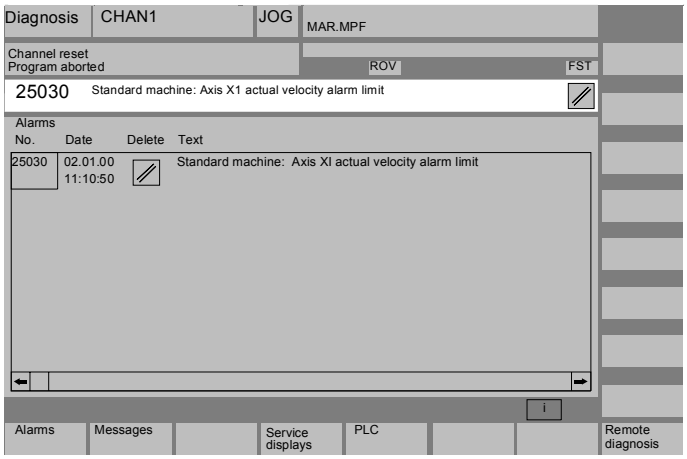
■

Diagnosis Operating Area

8.1 Basic display diagnosis

A display headed "Alarms" appears when you select the operating area "Diagnosis".

Basic screen



Explanation of display

Number	The alarm number is output under "Number". The alarms are output in chronological order.
Date	The time at which the alarm occurred is displayed with date, hour, minute, second, 100's second.
Delete criterion	The symbol denoting the alarm abort key is displayed for every alarm.
Text	The alarm text is displayed under "Text".

Horizontal softkeys

Alarms	All active alarms are displayed in the "Alarms" display.
Messages	An overview of active messages is displayed.
Service displays	You can view updated information about axes and drives installed in your system under softkey "Service displays".
PLC	Information on the current status of the PLC memory locations is displayed.
Remote diagnosis	If this option is set, it is possible to control and influence the operation of a control from a remote PC, as well as to transmit process data.

8.2 Displaying alarms and messages

Function

You can display a list of alarms and messages.

Operating sequence

Alarms:

Alarms

The alarm overview displays all active alarms with alarm numbers, date, clearance criteria and descriptions.

Clear the alarm by pressing the key that is displayed as a symbol:
Switch device off and on again (main switch)
or NCK POWER ON

Reset

Press "Reset" key.

ALARM CANCEL

Press "Alarm Cancel" key.

Cycle Start

Alarm is cleared with the "Cycle Start" key.

Alarm is cleared with the "Recall" key (message box).

Messages:

Messages

PLC operational messages that do not have to be acknowledged (as standard) (configurable).

Acknowledgement symbols:

You can use a machine datum to set which acknowledgement symbol is to be displayed for PLC alarms.
The following symbols are available:

PLC or

Display several alarms in succession:

By setting a machine datum, you can display several alarms (NCK, PLC, HMI) in succession in the alarm line. Each alarm remains visible in the set tool life until it is displaced by the next alarm.

8.3 Service display

8.3.1 Service axis

Function

The information in the "Service Axis" display is used to

- check the setpoint branch (e.g. position setpoint, speed setpoint, spindle speed setpoint prog.)
- check the actual-value branch (e.g. position actual value, measuring system 1/2, actual speed value), optimize the position control of the axis (e.g. following error, control difference, servo gain factor)
- check the entire control loop of the axis (e.g. through position setpoint/actual-value comparison and speed setpoint/actual-value comparison)
- check hardware faults (e.g. encoder check: If the axis is moved mechanically, the actual position value must change)
- set and check axis monitoring functions.

References /FB1/D1: Diagnostic tools

Operating sequence

Service displays

Select the menu headed "Service displays".
The horizontal softkey bars change.

Service axis

The "Service axis" window displays information about the machine axis together with axis name and axis number.

PAGE DOWN

You can page up and down with the "Page" keys.

Axis + Axis -

The service values of the next (+) and the previous (-) axis are displayed.

8.3.2 Service drive

Function

The information contained in the "Service drive" display is used to

- check the status on enabling and control signals (e.g. pulse enable, drive enable, motor selection, setpoint parameter set), and on FDD/MSD operating modes (e.g. setup mode, parking axis)
- display temperature warnings, check the current setpoint/actual-value display (e.g. position actual-value measuring system 1/2, speed setpoint, speed actual value)
- check the drive status
- display the current ramp-up phase
- display total error messages (Message ZK1), display the status messages of the drive (e.g. threshold torque not reached, actual speed = set speed)

References /FB1/D1: Diagnostic tools

Operating sequence

Service displays

Select the menu headed "Service displays".
The horizontal softkey bars change.

Service drive

The "Service drive" window displays information about the axis drive together with axis name and number.

PAGE DOWN PAGE UP

You can page up and down with the "Page" keys.

Drive + Drive -

The service values of the next (+) and the previous (–) drive are displayed.

8.3.3 Service safety integrated

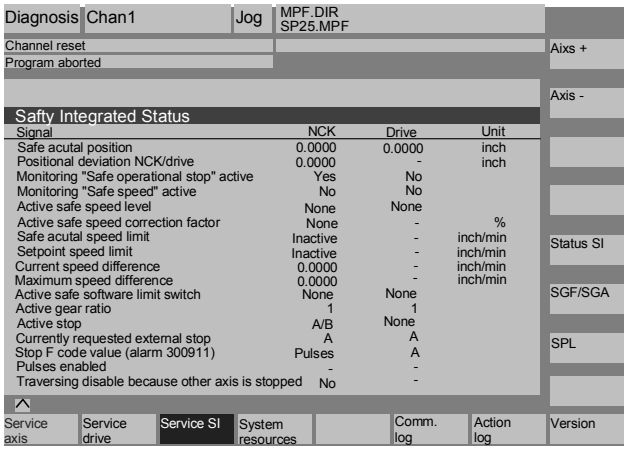
Function

Upon activation of the "Service SI" softkey, three information blocks about SI-related data are displayed on the HMI Embedded for the selected axis:

- Status SI (selected by default)
- SGE/SGA
- SPL

References: FBSI/, Description of Functions Safety Integrated.

Status SI



Set the required axis via the vertical softkeys "Axis +" and "Axis –". The active axis is displayed in the top right half of the table.

Available signals/values

Safe actual position
Positional deviation NCK/drive
"Safe operational stop" monitoring active
"Safe speed" monitoring active
Active safe speed level
Active safe speed correction factor
Safe actual speed limit
Safe setpoint speed limit
Current speed difference
Maximum speed difference
Active safe software limit switch
Active gear ratio
Active stop
Currently requested external stop
Stop F code value (alarm 300911)
Pulses enabled
Traversing disable due to stop on other axis

Operating sequence

Service displays

Select the menu headed "Service displays".
The horizontal softkey bars change.

Service SI

The "Service SI" window displays information about Safety Integrated data together with the associated axis name and axis number.

Axis +

Axis -

The service values of the next (+) and the previous (-) axis are displayed.

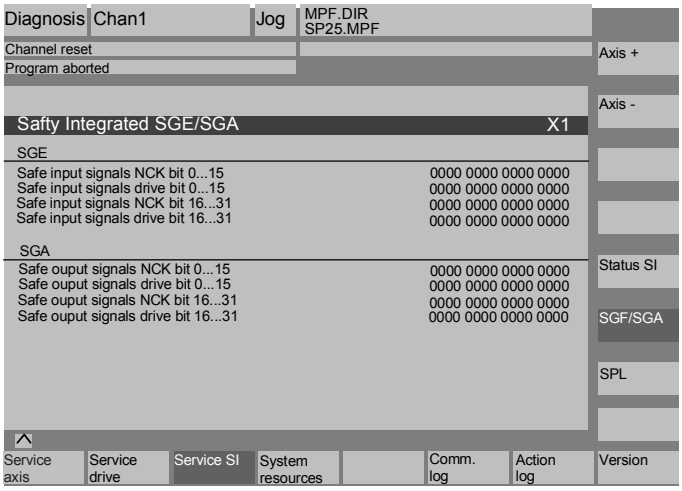
SGE/SGA

Use this softkey to display the safety-relevant input and output signals SGE and SGA.

SPL

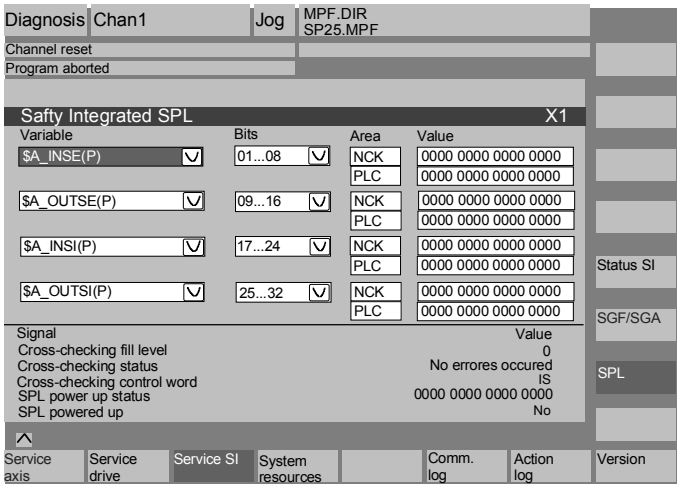
Use this softkey to display the safe programmable logic signals SPL.

SGE/SGA



The available signals can be seen in the above screen.
The vertical softkey Status SI takes you to the Status SI screen, SPL opens the Safe Programmable Logic screen.

SPL



In the "Variable" selection box, you can select:
$A_INSE(P) corresponds to simultaneous activation of
$A_INSE upper line, origin of the NCK and
$A_INSEP lower line, origin of the PLC
and effectively the same for the other variables:
$A_OUTSE(P)
$A_INSI(P)
$A_OUTSI (P)
$A_MARKERSI(P)

Under Bit you can request an 8-bit range from the selected signal.

Saving

The variables that have been selected and the associated bit areas are saved and are taken into account when subsequently selecting the screen.
In addition to the current values, the origin of the displayed NCK/PLC signals is displayed.

The settings are reset the next time the control is powered up.

Displayed signals

Cross-checking fill level (CDC Crosswise data comparison)
Cross-checking status
Cross-checking control word
SPL power up status
SPL powered up

SPL power up status: Bit0: SPL interface parameters set
Bit1: SPL program file SAFE.SPL loaded
Bit2: NCK waiting for PLC power-up
Bit3: PLC power-up complete
Bit4: Interrupt needs to be assigned for SPL start
Bit5: Interrupt has been assigned for SPL start

Bit6: Interrupt processing for SPL start called
Bit7: Interrupt processing for SPL start terminated
Bit9: NCK cross-checking has been started
Bit10: PLC cross-checking has been started
Bit11: Cyclic SPL checksum checking active
Bit12: All SPL protection features active

8.3.4 Display and modify system resources

Function

You can display the system resources currently in use for the NC and HMI Embedded areas (utilization display).

Further information

The display is also available in the "Start-up" operating area.
The procedure is described in Chapter 9, Section entitled "Display and Change System Resources".

8.3.5 Communications fault log

Function

Errors occurring in the communication between HMI Embedded and NCK/PLC are recorded in a communication error log.

Comm. Log

You can display this log by pressing the "Comm. Log." key.

The error log file is principally used by the control manufacturer (Siemens) as a diagnostic tool for communication errors.

The communication error log file can be read out via the RS-232 interface in the "Services" operating area.

8.3.6 Display version data

Function

You can read the HMI and the NCU version in the version window.
The HMI Embedded is also displayed in the run-up screen as well as during installation.
The complete product identifier (e.g. HMI-Embedded) also appears in front of the 8 digit version identifier.
In the "Services" operating area, the version data can be read from the (System, File "VERSION_SYF") directory via the RS-232 C interface or via one of the logical drives (NC-card).

Operating sequence

Service displays

Press "Service displays" softkey.
The horizontal softkey bars change.

Version

Open the "Version" window. More areas are available and you can see the version data via the following horizontal softkeys:

NCU Version

Version display of the NCU

HMI Version

Version display of the HMI Embedded.

Compile cycles

Only if option selected, version display for compile cycles.
see Chapter: "Displaying Loadable Compile Cycles"

Cycles version

Version display for cycles

PAGE DOWN PAGE UP

Use the "Page" keys to scroll up and down.

Save versions

Data are saved and can be read out in the "Services" operating area.

8.3.7 Displaying loadable compile cycles

Function

If compile cycles are available in the NCK, you can display them in a separate version display. Besides the current version (name, extension, type, date, time, length) and the start address (path/NC-card), the access authorization for "reading, writing, enabling, listing and deleting" are displayed.

Loadable compile cycles are available for the following software versions:

- NCK SW 6.3 and higher
- HMI Embedded: (SW 6.5 and higher)

Operating sequence

Service displays

Compile cycles are loaded when you initiate an NCU reset and can be displayed.

Version | Compile cycles

When you open the "Version" window in the "Service displays" menu, the "Compile cycles" softkey is displayed.

All loaded files of the type **.elf** are displayed in the "Version data compile cycles" overview.

In the "Services" operating area, you can output data via RS-232 C in the "Manage data" basic screen. Additional storage places, such as logical drives, are also possible.

Properties

Press the "Properties" softkey
The horizontal and vertical softkey menu changes and a "Properties" window, such as the one below for NC card, appears.

Version display

Features			
Path	:\NC-Card\Loadable-Compile-Cycles		
Name	:CCMCSC	Date:	Time
Extension	:ELF	Length:	Loaded: ☐
Type	:Loadable compile cycle		
Access authorization			
Reading:	Write:	Enable:	List: Delete:
Contents: Loadable compile cycle			
Version: MCSC Coupling axes MCS		Time	Date
Advance version of compile cycle (Preliminary)			
Interface: 001.001@Interfaces=002.000 @TChain=001.000			
Current access level:			

8.4 PLC

8.4.1 General

Function

The function is also available in the "Services" operating area.

You can obtain information about the current states of the following memory locations of the PLC and change them if necessary:

Inputs:	Input bit (Ex), input byte (EBx) Input word (Ewx), input double word (Edx)
Outputs:	Output bit (Ax), output byte (Abx) Output word (Awx), output double word (Adx)
Bit memories:	Memory bit (Mx), memory byte (MBx) Memory word (MWx), memory double word (MDx)
Timers:	Time (Tx)
Counters:	Counter (Cx)
Data:	Data block (DBx), data bit (DBxx), data byte (DBBx), data word (DBWx), data double word (DBDx)
Format:	B = binary H = hexadecimal D = decimal G = floating comma (for double words) C = character (ASCII character)

Operand	Example	Reading	Writing	Format	Value	Area
Inputs		Yes	Yes			0-127
	E 2.0			B	0	
	IB 2			B	0101 1010	
				H	5A	
				D	90	
Outputs		Yes	Yes			0-127
	Q20.1			B	1	
	QB 20			B	1101 0110	
				H	D6	
				D	214	
Markers		Yes	Yes			0-255
	M 60.7			B	1	
	MB 60			B	1101 0110	
	FW 60			H	B8	
				D	180	
				C		
Timers	T20	Yes	No			0-31
				B		
				H		
				D		

Operand	Example	Reading	Writing	Format	Value	Area
Counters	C20	Yes	Yes	B H D		0-31
Data block **Data byte**	DB3.DBB9	Yes	Yes	H D B C	Q 10 000 0000 0000 1010	0-255 0-255

A maximum of 10 operands can be displayed simultaneously.

Changes can only be made to the PLC operands with the appropriate password.

Danger

Changes in the states of PLC memory locations have a major impact on the machine. Incorrect parameterization can endanger lives and cause the destruction of the machine.

8.4.2 Change/delete operand value

Function

The values of operands can be changed.

Operating sequence

The function can also be selected in the "Services" operating area.

PLC

Press softkey "PLC".
The first operand screen form appears.
The vertical softkey bar changes.

Change

Cyclic updating of the values is interrupted.

Operand + Operand -

You can increase or decrease the address of the operand by 1 place at a time.

Delete

Delete:
The entries for the selected operand (formats and values) are deleted.
A query window is opened.

Abort

Undo changes:
Press the "Abort" softkey.
Cyclic updating is continued; the entered values are not transferred to the PLC.

Accept

Accept:
The entered values are transferred to the PLC. Cyclic updating is continued.

Further information

HELP

Press the "Information" key.
A description of the permissible input syntax for the PLC status display is overlaid.

8.4.3 Selecting/creating operand forms for PLC status

Function

You can save the operands entered in the "PLC status" window to a file or read in a back-up list of operands.
The values in the PLC status are automatically discarded when the system is restarted cold (after POWER ON).
As a way of optimizing the entry of operands and formats in the PLC status, you can read in operand screen forms from DOS files (one file for each screen form).
These screen forms are generated in a special syntax in ASCII format.

Naming conventions for DOS files:

name.plc	name is a screen form name of max. 8 characters

Content of the DOS file:

[\\Comment]	e.g.//form for the PLC test
Operand/Format	DB0.DBB0/B
[Operand/Format]	DB1.DBW0/H
.	.
.	.
[Operand/Format]	T100-D

You can enter as many comments and operand/format lines as you wish. In accordance with the PLC status display only the first 10 operands/format lines are read in.

Operating sequence

The function can also be selected in the "Services" operating area.

PLC

Press softkey "PLC".
The first operand screen form appears.
The vertical softkey bar changes.

Read in operands

Press softkey "Read in operands" (via application diskette).

OK

Position the cursor on the operand form you wish to find and confirm by pressing softkey "OK".
The screen form you selected is imported into the PLC status display.

8.4.4 Setting the time/date

Function

You can set the date and time on the PLC and synchronize the date and time between the PLC and HMI Embedded.

Operating sequence

PLC

Press the "PLC" softkey.
The horizontal and vertical softkey bars change.

Set date/ clock

Press the "Set date/clock" softkey. Open the window to enter the date and time.
Use the cursor key to move to the input fields of date (day/month/ year) and time (hour/min./second).

Enter the correct values in the fields and save these with the "Input" key.

Accept

By pressing "Accept", the date and time of HMI Embedded are transferred to the PLC.
You can check the synchronization in the "Current:" output field.

References

/IAM/ IM2: HMI Installation & Start-up Guide,

The values set are retained when the software is rebooted.

Further information

Sync. manual

You can also manually enter the time period for synchronization of the HMI and PLC clock via softkey "Sync. manual".

Sync. AUTO/ON

You can activate cyclic (automatic) clock synchronization via the "Sync. AUTO/ON" softkey and set the time duration for synchronization there. Enter the correct values in the input fields.

Sync. AUTO/OFF

Automatic synchronization is deactivated by pressing the "Sync. AUTO/OFF" softkey. The HMI and PLC then run separately again.

Machine manufacturer Always refer to the information and specifications provided by the machine manufacturer.

8.5 Activate remote diagnosis

The remote diagnosis function is an option.
If this option is set, it is possible to control and influence the operation of a PCU 20 from a remote PC, as well as to transmit data.

Remote diagnosis

When you select the "Remote diagnosis" softkey, the vertical "Start" and "End" softkeys appear. In this way you start or end the remote diagnosis.

References /IAM/ IM2: Installation & Start-up Guide HMI Embedded

Machine manufacturer Always refer to the information and specifications provided by the machine manufacturer.

■

Start-Up Operating Area

9.1 Start-up basic display

Danger

Changes in the startup operating area have a significant influence on the machine. Incorrect parameterization can endanger lives and cause the destruction of the machine.

Access rights to certain menus in the startup operating area may be blocked by means of a keyswitch or password.

The functions described can be executed by the machine operator with his access rights.

For more detailed information on the subject of startup for

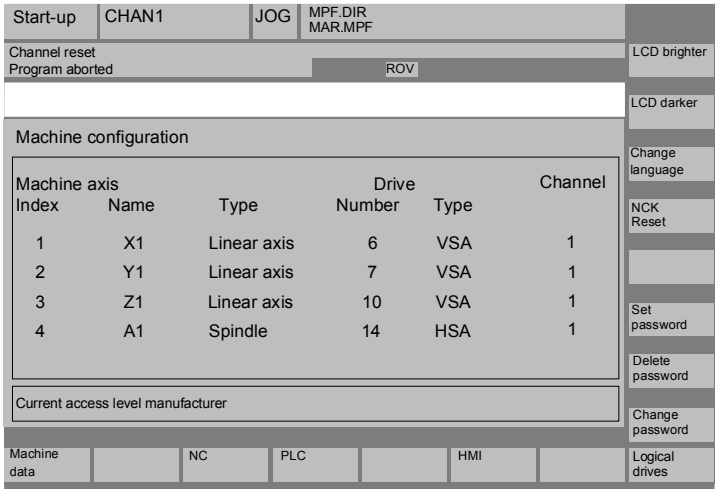
- System personnel
- Machine manufacturer
- Service personnel
- Machine users (machine setters)

Please refer to the following documentation:

References

/IAD/Start-up Guide, SINUMERIK 840D
/IAC/Start-up Guide, SINUMERIK 810D
/IAM/IM2, Installation & Start-Up Guide HMI Embedded

The "Machine configuration" window is displayed in the "Start-up" basic display.



Horizontal softkeys

Machine data	Enables you to change the machine data for all areas.
NC	Here you can look at the NC settings for NC power up, the NC address and NCU switchover and change them if necessary.
PLC	The function PLC status is available. You can update date and time of the PLC and HMI.
HMI	Here you can enter the basic settings for the operator panel front (e.g. color settings).
Logical drivers	You can select the configuration screens for the new network by activating the "Logical drives" softkey. The "Logical drives" can refer to either a network connection or an internal drive, e.g. a disk drive, a compact flash card etc.

Vertical softkeys

LCD brighter / LCD darker	Only for OP 010 with STN display: You can change the screen brightness.
Change language	You can use two languages in parallel. When you select softkey "Change language", the screen text is displayed in the other language.
	If the language is not loaded, "?" is output. When you select softkey "Change language" again, the text display reverts back to the other language.
NCK Reset	You can press this softkey to initiate NCK power ON/Reset.
Set password	You can set a password.
Delete password	You can delete a password.
Change password	You can change your password.

9.2 Display machine data

Access to the Machine data operating area can be controlled by key switch or password.

Function

Areas

The machine data are divided into the following areas:

1. General machine data ($MN)
2. Channel-specific machine data ($MC)
3. Axis-specific machine data ($MA)
4. Drive-specific machine data ($MD)
5. Display machine data ($MM)

A separate list display in which you can view and change machine data is provided for each of these areas.

The following information about the machine data is displayed from left to right:

- Machine data number
- Machine data name (without area identification `$MN` , `$MC` , `$MA` , `$MD` , `$MM`), possibly with field index.
- Value of the machine datum.
- Unit of the machine datum.
- Effectiveness

If the machine data do not use units, no units are displayed.
If data is not available, the "#" symbol is displayed instead of the value.
If the value ends in an "H", it is a hexadecimal value.

The physical units of machine data are displayed on the right-hand side of the input field.

Examples:

m/s**2	m/s^2 (meters/second squared): acceleration
rev/s**3	rev/s^3 (revolution/second to power of 3): Change in rate of acceleration for rotating axis
kg/m**2	kg/m^2 (kilogram/meters squared): Moment of inertia
mH	mH (millihenry): Inductance
nm	Nm (Newton meters): Torque
us	µs (microseconds): Time
µA	µA (microamperes): Amperage
µVs	µVs (microvolt-seconds): Magnetic flux

userdef User-defined: The unit is defined by the user.

The abbreviation in the right-hand column indicates the activation criterion for a machine data:

- so = effective immediately
- cf = via "MD set effective" softkey activation
- re = Reset
- po = POWER ON (NCK Power On Reset)

References

/IAD/Installation & Start-up Guide: 840D
/IAC/Installation & Start-up Guide: 810D
/IAF/Installation & Start-up Guide: FM-NC

Operating sequence

Machine data

Pressing the "machine data" softkey changes the horizontal and vertical softkey bars.
You can select the machine data area you want by pressing the following softkeys:

General MD

- General machine data ($MN_),

Channel-spec. MD

- Channel-specific machine data ($MC_),

Axis-specific MD

- Axis-specific machine data ($MA_).

Drive configuration

In the "Drive configuration" menu you can find information about the drive modules that were configured via the startup tool or change the drive configuration.

Danger

Changes in the configuration data have a considerable influence on the machine. Incorrect parameterization can endanger lives and cause the destruction of the machine.

Drive MD

Drive-specific machine data ($MD_)

Display MD

Operator panel front machine data ($MM_)

FDD

Machine data for the feed drive,

MSD

Machine data for the main spindle drive,

Vertical softkeys

Find...

To find a specific machine datum, press the "Find…" softkey. Enter the name or number of the machine data you are looking for and press the "OK" softkey.

Keep searching

After pressing the "Keep searching" softkey, you move from one datum to the next in the machine data list.

9.2.1 Display options: Masking filter

Function

The purpose of masking filters is to selectively reduce the number of displayed machine data. For this function, all machine data in areas

- General machine data
- Channel-specific machine data
- Axis-specific machine data
- Drive configurations

are assigned to specific groups (e.g. configuration data, etc.).
The following applies:

1. Each area has its own group organization.
2. Each group corresponds to one bit in the filter word ("spare" bit in previous SW)
3. Each area has a maximum of 13 groups (group 14 is reserved for Expert mode (see below), bit 15 is reserved for expansions).

Display machine data do not have any group organization.

Filter criteria

The following table shows the criteria for displaying machine data in the order in which they are evaluated:

Criterion	Check
1. Access authorization	If the level of access authorization is not sufficient, the MD is not displayed. Otherwise criterion 2 is checked.
2. Masking filter active	The MD is always displayed when the filter is not active. Otherwise criterion 3 is checked.
3. Expert mode	The MD is not displayed if the expert mode bit is set and expert mode is not selected. Otherwise criterion 4 is checked.
4. Groups	If at least one group bit is both set and selected in the masking filter, criterion 6 is checked. Otherwise criterion 5 is checked.
5. All others	If none of the group bits is set and "All others" is selected in the masking filter, then criterion 6 is checked. If none of the group bits is set and "All others" is not selected in the masking filter, then the MD is not displayed.
6. Index from to	If the index check is selected and the index of an array is within the chosen range, then the MD is displayed. If the index check is selected and the index of an array is not within the chosen range, then the MD is not displayed.

Initialization

When you open a machine data window, the filter setting that matches the area is automatically updated.

Storing filter settings

Machine manufacturer Please see information supplied by machine tool manufacturer.

Operating sequence

Machine data

Press the "Machine data" softkey.
The horizontal and vertical softkey bars change.

Display options

Press the "Display options" softkey. A list of all the ranges that can be displayed/hidden appears.

Vertical softkeys

Select all

Press the "Select all" softkey and all areas are selected.

Deselect all

Press the "Deselect all" softkey and no areas are selected.

SELECT

To select individual areas, jump to the individual fields with the direction key and choose these fields with the Select key.

OK

Press "OK" to save your selection.

Abort

"Abort" takes you back to the previous screen.

9.3 NC settings

Function

NC

Here you can look at the NC settings for NC power up, look at and change the NC address if necessary, and carry out an NCU switchover.

Setting a key position for the startup switch with appropriate access rights.

Start up switch

After pressing the "Start up switch" softkey, you can choose from the following:

- ☐ Normal power-up
- ☐ Start-up with default values (Start-up mode)
- ☐ Software updates start

NCK address

Look at the NCK address and change it.
In general, address changes are only necessary for M to N operation.
You cannot undo an address change with general reset. The address change is not saved in an NC series startup archive.

OK or Save

If you press "OK" or "Save", the following safety query appears:
"Do you want to change the NCK address?
The changes will not become effective until an NCK Power on Reset has been performed".
Confirm with "Yes" or abort with "No".

Abort

"Abort" takes you back to the previous screen without saving.

9.4 PLC settings

Function

PLC

The following functions are available via the "PLC" softkey.

- Setting the time and date
- Importing operands.

Changes can only be made to the PLC operands with the appropriate password.

The procedure for handling PLC operands is described in the "PLC" section in Chapter 8, Diagnosis Operating Area.

Danger

Changes in the states of PLC memory locations have a major impact on the machine. Incorrect parameterization can endanger lives and cause the destruction of the machine.

9.4.1 Setting the time and date

Function

Set date/ time

You can set the date and time on the PLC and synchronize the date and time between the PLC and HMI Embedded.

The procedure is already described in Chapter 8, Diagnosis Operating Area, Section on "Setting Date/Time".

9.4.2 Import operands

Function

Import operands

For optimizing the input of operands and formats in the PLC status, you can import operand screens from DOS files.

The procedure for handling PLC operands is described in the "PLC" section in Chapter 8, Diagnosis Operating Area: "Selecting/Creating Operand Forms for PLC Status".

9.5 Change HMI surface

Function

You can make the following individual settings to your HMI.

- Color settings on screen
- Specify language selection
- Switch the logging process on (traverse log)
- Open ASCII editor

Operating sequence

HMI

Pressing the "HMI" softkey changes the horizontal and vertical softkey bars.
The following **submenus** are available via the horizontal softkeys:

9.5.1 Setting colors

Colors

If you press the "Colors" softkey, the "Color settings" menu appears:
You can choose from the color scheme:

- Standard (cannot be changed)
- Customized.

When you select the "Customized" color setting, you can change the color settings for the HMI user interface (e.g. background, window frames, text).

Save

Press "Save" and the current color setting is saved and you

OK

return to the menu by pressing "OK".

9.5.2 Traverse log

Traverse log

When you press the softkey, a screen is opened for setting parameters and activating the action log. You can make and save the following settings:

- Switch the logging process ON/OFF
  Log on:
- Select the data you want to log:
  Interrupts
  Keys
  channel status/override,
  Windows IDs

OK

You can save the settings via the vertical softkeys.

9.5.3 Open ASCII editor

Editor

For service only!
Go to the ASCII editor to display the files at DOS level.

DOS SHELL

For service only!
Go to the DOS level.

9.6 Set up logical drives

Function

Connections for a Compact Flash Card, Floppy, and network can be used on the PCU 20 by means of logical drive definitions.
This "Logical drive" function is the "Manage network/disk drive on the PCU 20" option.

Logical drives

In addition to the "NC" softkey, via which the NC main memory data can be displayed, a further 4 connections can be set up. These user-configured softkeys can be used to display directories and programs e.g. on disks, network drives and compact flash cards, or in the OEM memory.

These softkeys are available in the "Program" and "Services" operating areas.

The procedure used to set up the additional softkeys has already been described.

References /IAM/IM2, Installation Guide HMI Embedded, Chapter 3 "Network Configuration Data, Connections: Logical Drives"

9.7 Change language

Function

Change Language

In principle, you can use two languages in parallel.
German, English, French, Italian and Spanish are available as standard. You can install even more languages via an additional Language CD.
The language switch is done with the "Change Language" softkey.
However, if you want to change the two languages, proceed as follows:

Operating sequence

Change the language default

Languages

With more than two installed languages, you can change the language setting via the "Languages" softkey (the standard languages are English, Spanish, French, and Italian).
Choose the two languages you require with the cursor key.

Confirm your choice with the "Select" selection key.

Confirm with "OK".

OK

References For installation and configuration of languages, see:
IAM/IM2, Chapter:. Software Installation/Upgrade

Language Selection

If you are using more than two languages, you can select the desired language by pressing the "Language Selection" softkey.

9.8 Initiate NCK reset

Function

NCK Reset

After pressing the "NCK Reset" softkey, the safety query "Do you want to initiate an NCK reset?" appears. If you press the "Yes" softkey the computer is restarted.
If you press the "No" softkey, you will return to the previous screen.

9.9 Setting, deleting, changing password

Function

The control has a protection level system for enabling data areas.
There are access levels 0 to 7;

- 0 is the highest and
- 7 is the lowest.

Protection levels

- 0 to 3 is controlled by means of passwords and
- 4 to 7 by means of key switch settings.

The operator only has access to information protected by one particular level and the levels below it. The machine data is assigned different protection levels as a standard measure.

Protection level	Locked by	**Area**
0	Password	Siemens
1	Password	Machine manufacturer
2	Password	Installation engineer, service
3	Password	End user
4	Key switch position 3	Programmer, machine setter
5	Key switch position 2	Qualified operator
6	Key switch position 1	Trained operator
7	Key switch position 0	Semi-skilled operator

Depending on the authorization level, it will be possible to edit data such as cycles and machine data.
You can alter the set password using the "Password" function.

If one of the above passwords is set, the keyswitch position is ignored.

Operating sequence

Set password

You can set a password via the vertical softkeys,

Delete password

delete a password and

Change password

change it.

9.10 Display or modify system resources

Function

You can display the system resources currently in use for the NC and HMI Embedded areas (utilization display).

Operating sequence

>

By selecting the Etc key, you can access the "NC memory" softkey with the appropriate access level.

NC memory

An overview of the current memory allocation appears when you press the softkey.

The "Memory overview" window shows the user memory,

- static user memory (SRAM) and
- dynamic user memory (DRAM)

with:

- Entire memory
- Used memory
- Free memory

displayed in bytes.

Further information

SRAM or DRAM

Further softkeys are available for displaying more detailed information about the memory capacities.

Details

In these individual windows, you can optimize the memory utilization by changing the machine data directly.

References

/IAM/IM2, Installation & Start-Up GuideHMI Embedded, Chapter 5: Displaying and Editing System Resources

■

Service

10.1 Operating data

Operating data

	Value
Air humidity, humidity class to DIN 40040	F
Atmospheric pressure	860 to 1080 hPa
Shock protection, Protection class to DIN VDE 0160	I
Degree of protection according to DIN 40050	
• Operator panel (front)	IP 54
• Operator panel front (rear)	IP 00
• Machine control panel (front)	IP 54
• Machine control panel (rear)	IP 00

You can find a complete summary of operating data on the appropriate supplementary sheets or in the documentation:

References /BH/ Operator Components, Manual

10.2 Cleaning

Cleaning agents

The front of the monitor and the surface of the operator panel front can be cleaned. For dirt that is relatively easy to remove, standard household dish washing liquid, or an industrial cleaner (such as "Special Swipe") can be used. These cleaners will also remove dirt containing graphite.

Cleansing agents which contain one or more of the following ingredients can be used for a short period of time:

- Thinned mineral spirits
- Bases
- Organic hydrocarbons
- Dissolved detergents

Plastic material used

The plastic material used on the front of the SINUMERIK 840D is suitable for applications on machine tools.

They are resistant to

1. Grease, oil, mineral oils
2. Bases and alkalis
3. Dissolved detergents and
4. Alcohol

Solvents such as chlorinate hydrocarbons, benzene, esters and ethers should be avoided!

■

Appendix

A Abbreviations

AS	Automation System
ASCII	American Standard Code for Information Interchange
ASIC	Application-Specific Integrated Circuit
ASUB	Asynchronous subprogram
AuxF	Auxiliary Function
BCD	Binary Coded Decimals
BCS	Basic Coordinate System
BIN	Binary files
BIOS	Basic Input Output System
BOT	Boot Files: for SIMODRIVE 611 D
BP	Basic Program
C1 ... C4	Channel 1 to Channel 4
CAD	Computer-Aided Design
CAM	Computer-Aided Manufacturing
CNC	Computerized Numerical Control
COM	Communication
COR	Coordinate Rotation
CP	Communications Processor
CPU	Central Processing Unit
CR	Carriage Return
CRC	Cutter Radius Compensation

CRT	Cathode Ray Tube
CSB	Central Service Board: PLC module
CSF	Control System Flowchart
CTS	Clear To Send (serial data interfaces)
CUTOM	Cutter radius compensation (tool radius compensation)
DAC	Digital-to-Analog Converter
DB	Data Block in the PLC
DBB	Data Block Byte in the PLC
DBW	Data Block Word in the PLC
DBX	Data Block Bit in the PLC
DC	Direct Control: The rotary axis is moved along the shortest path to the absolute position within one revolution.
DCD	Data Carrier Detect
DCE	Data Communications Equipment
DDE	Dynamic Data Exchange
DIN	Deutsche Industrie Norm (German Industry Standard)
DIO	Data Input/Output: Data transfer display
DIR	Directory
DLL	Dynamic Link Library
DOS	Disk Operating System
DPM	Dual-Port Memory
DPR	Dual-Port RAM
DRAM	Dynamic Random Access Memory

DRF	Differential Resolver Function
DRY	Dry Run
DSB	Decoding Single Block
DTE	Data Terminal Equipment
DW	Data Word
EIA Code	Special tape format: Number of perforations per character is always odd
ENC	Encoder actual-value sensor
EPROM	Erasable Programmable Read Only Memory
ERROR	Error from printer
FB	Function block
FBS	Slimline screen
FC	Function Call: Function block in PLC
FCI	Free Contour Input
FDB	Product Database
FDD	Floppy Disk Drive
FEPROM	Flash EPROM Read/write memory
FIFO	First-In-First-Out: Memory which operates without address specification from which data are read in the same order as they are stored.
FIPO	Fine Interpolator
FM	Function Module
FM-NC	Function module – numerical control
FPU	Floating Point Unit

FRA	Frame Block
FRAME	Data block (frame)
FSD	Feed Spindle Drive
FST	Feed Stop
GUD	Global User Data
GWPS	Grinding Wheel Peripheral Speed
GWRC	Grinding Wheel Radius Compensation
HD	Hard Disk
HEX	Abbreviation for hexadecimal
HHU	Hand-Held Unit
HMS	High-Resolution Measuring System
HW	Hardware
I	Input
I/O	Input/output
I/RF	Infeed/Regenerative Feedback Unit (power supply) of SIMODRIVE 611(D)
IBN	Start-Up
IC (GD)	Implicit Communication (Global Data)
ICA	Interpolative Compensation with Absolute values
IM	Interface Module
IMR	Interface Module Receive
IMS	Interface Module Send
INC	Incremental dimension

INI	Initializing Data
IPO	Interpolator
IS	Interface Signal
ISA	International Standard Architecture
ISO	International Standard Organization
ISO Code	Special tape code, number of perforations per character is always even
JOG	Jog mode: Setup mode
K_{UE}	Transmission ratio
K_V	Servo gain factor
LAD	Ladder diagram (programming method for PLC)
LCD	Liquid-Crystal Display
LEC	Leadscrew Error Compensation
LED	Light Emitting Diode
LF	Line Feed
LUD	Local User Data
MB	Megabyte
MC	Measuring Circuit
MCP	Machine Control Panel
MCS	Machine (Machine Coordinate System)
MD	Machine Data
MDA	Manual Data Automatic: Manual input
MLFB	Machine-readable product designation (= Order No.)

MMC	Man Machine Communication: SINUMERIK operator functionality for operation, programming and simulation. MMC has the same meaning as HMI.
MPF	Main Program File: NC parts program
MPI	Multi Point Interface
MS	Microsoft (software manufacturer)
MSD	Main Spindle Drive
NC	Numerical Control
NCK	Numerical Control Kernel (with block preparation, traversing range, etc.)
NCU	Numerical Control Unit: Hardware unit of the NCK
NRK	Name for the operating system of the NCK
NURBS	Non-Uniform Rational B-Spline
O	Output
OB	Organization Block in PLC
OEM	Original Equipment Manufacturer
OI	Operator Interface
OM	Operating Mode
OP	Operator Panel
OPI	Operator Panel Interface
OPT	Options
OSI	Open System Interconnection
PC	Position Control
PCIN	Name of the software for data communication with the control

PCMCIA	Personal Computer Memory Card International Association: Memory card standardization
PE	Pulse enable for drive module
PG	Programming Device
PLC	Programmable Logic Controller
PMS	Position Measuring System
POS	Positioning
RAM	Random Access Memory (read-write memory)
REF	Reference point approach function
REPOS	Reposition function
RISC	Reduced Instruction Set Computer: Processor type with small instruction set and high-speed instruction throughput
ROV	Rapid Override
RPA	R Parameter Active Memory area in NCK for R variable numbers
RPY	Roll Pitch Yaw: Type of rotation of a coordinate system
RS-232	Serial Interface (US standard), defines transmission of serial data between DTE and DCE devices
RTS	Request To Send (serial data interfaces)
SBL	Single Block
SCK	Software Configuration Kit
SD	Setting Data
SDB	System Data Block
SEA	Setting Data Active: (file identifier for setting data)
SFB	System Function Block

SFC	System Function Call
SK	Softkey
SKP	Skip block
SM	Stepper Motor
SPF	Sub Program File: Subroutine
SR	Subroutine
SRAM	Static RAM (battery-backed)
SSI	Serial Synchronous Interface
STL	Statement list
SW	Software
SYF	System Files
T	Tool
TC	Tool Change
TEA	Testing Data Active: Identifier for machine data
TLC	Tool Length Compensation
TNRC	Tool Nose Radius Compensation
TO	Tool Offset
TOA	Tool Offset Active: Identifier (data type) for tool offsets
TRANSMIT	Transform Milling into Turning: Coordinate conversion on turning machines for milling operations
TRC	Tool Radius Compensation
UFR	User Frame: Zero offset
UI	User Interface

WCS	Work (Workpiece Coordinate System)
WO	Work Offset
WOA	Work Offset Active: Identifier (file type) for zero offset data
WOP	Workshop-Oriented Programming
WPD	Work Piece Directory
µC	Micro Controller

B Terms

Key terms are given in alphabetical order. Terms which appear in the explanatory part and for which there is a separate entry are referred to with the "->" symbol.

A

Absolute dimensions

A destination for an axis movement is defined by a dimension that refers to the origin of the currently active coordinate system. See also -> incremental dimension.

Acceleration with jerk limitation

In order to optimize the acceleration response of the machine whilst simultaneously protecting the mechanical components, it is possible to switch over in the machining program between abrupt acceleration and continuous (jerk-free) acceleration.

Access authorization

The CNC program blocks and data are protected via a 7-stage access authorization procedure.

- three password levels for system manufacturers, machine manufacturers and users, plus
- four keyswitch positions which can be evaluated via the PLC.

Address

An address is the identifier for a certain operand or operand range, e.g. input, output etc.

Analog input/output module

Analog input/output modules are signal transducers for analog process signals.
Analog input modules convert analog measured values into digital values which can be processed in the CPU.
Analog output modules convert digital values into analog output signals.

Approach machine fixed-point

Approach motion towards one of the predefined -> fixed machine points.

Archiving

Reading out data and/or directories to an **external** memory device.

A-Spline

The Akima-Spline runs under a continuous tangent through the programmed interpolation points (3rd order polynomial).

Asynchronous subroutine

A parts program which can be started asynchronously to (independently of) the current program status by an interrupt signal (e.g. "rapid NC input" signal).

Automatic Operating mode of the control (block sequence operation according to DIN): Operating Mode in NC systems in which a -> parts program is selected and continuously executed.

Auxiliary functions Auxiliary functions can be used to transfer -> parameters to the -> PLC in -> parts programs, where they trigger reactions which are defined by the machine manufacturer.

Axes In accordance with their functional scope, the CNC axes are subdivided into:

- Axes: interpolating path axes
- Auxiliary axes: non-interpolating feed and positioning axes with an axis-specific feed rate. Auxiliary axes are not involved in the actual machining, and include for example tool feeders and tool magazines.

Axis address See -> axis identifier

Axis identifier Axes are labeled in accordance with DIN 66217 (for a clockwise orthogonal -> coordinate system) with the letters X,Y, Z.
-> Rotary axes rotating around X, Y, Z are assigned the identifiers A, B, C. Additional axes, which are parallel to those specified, can be identified with other letters.

Axis name See -> axis identifier

B

Backlash compensation Compensation for mechanical machine backlash, e.g. backlash on reversal for feed screws. Backlash compensation can be entered separately for each axis.

Backup Saving the memory contents to an external memory device.

Backup battery The backup battery ensures that the -> user program is reliably backed up in the -> CPU against mains failure and that fixed data areas and markers, times and counters are kept in non-volatile memory.

Back-up memory The backup memory enables buffering of memory areas of the -> CPU without a buffer battery. Buffering can be performed for a configurable number of times, counters, markers and databytes.

Basic axis Axis whose setpoint or actual value position forms the basis of the calculation of a compensation value.

Basic coordinate system Cartesian coordinate system which is mapped by transformation onto the machine coordinate system.
In the -> parts program, the programmer uses the axis names of the basic coordinate system. The basic coordinate system exists in parallel to the -> machine coordinate system when no -> transformation is active. The difference between the systems relates to the axis identifiers.

Baud rate Rate of data transfer (Bit/s).

Blank Workpiece as it is before a part is machined.

Block "Block" is the term given to any files required for creating and processing programs.

Block search For testing part programs or following interruption of machining, it is possible to select any point in the part program using the block search function in order to start or resume at this point.

Booting Loading the system program after power on.

B-spline With the B-spline, the programmed positions are not interpolation points, as they are just "control points" instead. The generated curve only runs near to the control points, not directly through them (optional 1st, 2nd or 3rd order polynomials).

Bus connector A bus connector is an S7-300 accessory that is supplied with the -> I/O modules. The bus connector expands the -> S7-300 bus from the -> CPU or an I/O module to the neighboring I/O module.

C

C axis Axis around which the tool spindle describes a controlled rotational and positioning movement.

Channel A channel is characterized by being able to run independently of other channels or a part program. A channel exclusively controls the axes and spindles assigned to it. Parts programs run on various channels can be coordinated by -> synchronization.

Channel structure The channel structure enables the -> programs of the individual channels to be executed simultaneously and asynchronously.

Circle interpolation The -> tool is required to travel in a circle between defined points on the contour at a specified feedrate while machining the workpiece.

CNC -> NC

CNC programming language The CNC programming language is based on DIN 66025 with high-level expansions. The -> high-level CNC language and programming allow, among other things, macros to be defined (groupings of individual instructions).

COM Component of the NC control for the implementation and coordination of communication.

Compensation axis Axis with a setpoint or actual value modified by the compensation value

Compensation table Table containing interpolation points. It provides the compensation values of the compensation axis for selected positions on the basic axis.

Compensation value Difference between the axis position measured by the position sensor and the desired, programmed axis position.

Continuous-path mode The purpose of continuous-path mode is to prevent excessive deceleration of the -> path axes at the part program block boundaries (in terms of the control, machine and other properties of the operation and the user) and to effect the transition to the next block at as uniform a path speed as possible.

Contour Outline of a -> workpiece.

Contour monitoring The following error is monitored within a defined tolerance band to ensure contour precision. An impermissibly high following error might be caused by a drive overload, for example. In this case an alarm is triggered and the axes are stopped.

Coordinate system See -> machine coordinate system, -> workpiece coordinate system

CPU Central Processor Unit, -> Programmable Logic Controller

C-spline The C-spline is the most well-known and widely used spline. The transitions at the interpolation points are continuous, both tangentially and in terms of curvature. 3rd order polynomials are used.

Cutter radius compensation Contour programming assumes that the tool is pointed. Because this is not actually the case in practice, the curvature radius of the tool used must be communicated to the control which then takes it into account. The curvature center is maintained equidistantly around the contour offset by the radius of curvature.

Cycle
Protected subroutine for executing a repetitive machining process on the -> workpiece.
An NC code generated via "Expand operator interface" or a geometry process consists of several lines.

Cycle support
In the "Program" operating area, the available cycles are listed under the menu "Cycle Support". After selecting the desired machining cycle the required parameters for the value assignment are displayed in clear text.

D

Data block
1. Data unit of the -> PLC, which the -> HIGHSTEP programs can access.
2. Data unit of the -> NC: Data blocks contain data definitions for global user data. These data can be initialized directly when they are defined.

Data transmission program PCIN
PCIN is an auxiliary program which is used to send and receive CNC user data via the serial interface, such as e.g. parts programs, tool offsets etc. The PCIN program can be executed under MS-DOS on standard industrial PCs.

Data word
A data unit, two bytes in size, within a -> data block.

Diagnostics
1. Control operating area
2. The control has both a self-diagnostics program and testing aids for service. Status, alarm and service indicators.

Digital input/output module
Digital modules are signal transducers for binary process signals.

Dimensions in metric units and inches
Position and gradient values can be entered in the machining program in inches. The control can be set to a basic system regardless of the programmed measuring system (G70/G71).

DRF
Differential Resolver Function: An NC function, which generates an incremental zero offset in automatic mode in conjunction with an electronic handwheel.

Drift compensation
During the constant traversing phase of the CNC axes an automatic drift compensation is performed for the analog speed control.

Dynamic feedforward control

Inaccuracies in the contour caused by following errors can be almost entirely eliminated with the aid of dynamic, acceleration-dependent feedforward control. The result is extraordinary machining precision even at high tool path feed-rates. The feedforward control can be individually selected and deselected for each axis in the parts program.

E

Editor

The editor is used to create, modify, add to, compress, and insert programs/texts/program blocks.

Electronic handwheel

The electronic handwheels can be used to simultaneously traverse selected axes manually. The meaning of the lines on the handwheels is defined by increment weighting.

Exact stop

With a programmed exact stop instruction, the position stated in a block is approached precisely and very slowly, if necessary. In order to reduce the approach time, -> exact stop limits are defined for rapid traverse and feed.

Exact stop limit

When all path axes reach their exact stop limits, the control responds as if it had reached its destination point precisely. The -> part program continues execution at the next block.

External work offset

Zero offset specified by the -> PLC.

F

Fast retraction from contour

When an interrupt arrives a motion can be initiated via the CNC machining program which allows the tool to be quickly retracted from the workpiece contour currently being machined. The retraction angle and the distance retracted can also be parameterized. An interrupt routine can also be executed following the fast retraction of the tool. (SINUMERIK FM-NC, 840D).

Feed override

The programmed velocity is overridden by the current velocity setting via the machine control panel or by the PLC (0-200 %). The feedrate can also be corrected by a programmable percentage factor (1 - 200%) in the machining program.

Finished-part contour

Contour of the finished workpiece. See also -> blank.

Fixed machine point

A point defined uniquely by the machine tool, e.g. the reference point.

Fixed-point approach Machine tools can approach fixed points such as a tool change point, loading point, pallet change point, etc. in a defined way. The coordinates of these points are stored in the control. Where possible, the control moves these axes in -> rapid traverse.

Frame A frame is an arithmetic rule that transforms one Cartesian coordinate system into another Cartesian coordinate system. A frame contains the components -> zero offset, -> rotation, -> scaling, -> mirroring.

G

Geometry Description of a -> workpiece in the -> workpiece coordinate system.

Geometry axis Geometry axes are used to describe a 2- or 3-dimensional range in the workpiece coordinate system.

Global main program/subroutine Every global main program/subroutine can only appear once under its own name in the directory, and it is not possible to have the same program name in different directories with different contents as a global program.

Ground Ground is taken as the total of all linked inactive parts of a device which will not become live with a dangerous contact voltage even in the event of a malfunction.

H

Helical interpolation Helical interpolation is especially suitable for easy machining inside or outside threads with form cutters and for milling lubrication grooves. The helix consists of two motions:
1. A circular movement in one plane
2. A linear movement perpendicular to this plane

High-level CNC language The high-level language offers: -> user variable, -> pre-defined user variable, -> system variable, -> indirect programming, -> mathematical and trigonometric functions, -> comparison operations and logic operations, -> program jumps and program branching, -> program coordination (SINUMERIK 840D), -> macro technology.

High-speed digital inputs/outputs The digital inputs can be used for example to start fast CNC program routines (interrupt routines). The digital CNC outputs can be used to trigger fast, program-controlled switching functions. (SINUMERIK 840D).

HIGHSTEP Summary of the programming options for the -> PLC in the AS300/AS400 system.

Home position Point on the machine tool used to reference the measuring system of the -> machine axes.

I

I/O module I/O modules represent the link between the CPU and the process. I/O modules are:
- ->Digital input/output modules
- ->Analog input/output modules
- ->Simulator modules

Inch system Dimension system which defines distances in inches and fractions of inches.

Inclined surface machining Drilling and milling operations on workpiece surfaces that do not lie in the coordinate planes of the machine can be performed easily using the function "inclined-surface machining".

Increment Traversed distance information via the number of increments. The number of increments can be stored as -> setting data or selected using keys labeled with 10, 100, 1000, 10 000.

Incremental dimension Also incremental dimension: A destination for axis traversal is defined by a distance to be covered and a direction referenced to a point already reached. See also -> absolute dimension.

Initialization block Initialization blocks are special -> program blocks. They contain value assignments that are performed before program execution.
The primary purpose of initialization blocks is to initialize predefined data or global user data.

Initialization files It is possible to create an initialization file for each -> workpiece. Various variable assignments which are intended to apply specifically to one workpiece can be stored in this file.

Interconnecting cables Connecting cables are pre-assembled or user-assembled 2-wire cables with a connector at each end. These interconnecting cables connect the -> CPU via the -> multipoint interface (MPI) with a -> programming device or other CPUs.

Intermediate blocks

Traversing movements with tool offset selected (G41/G42) can be interrupted by a limited number of intermediate blocks (block without axis movements in the compensation plane) whereby the tool offset can still be correctly calculated. The permissible number of intermediate blocks which the control reads ahead can be set in system parameters.

Interpolator

Logical unit of the -> NCK which determines intermediate values for the movements to be traversed on the individual axes on the basis of destination positions specified in the parts program.

Interpolatory compensation

Interpolatory compensation can be used to compensate for leadscrew errors and measuring system errors (LEC, MSEC) caused during production.

Interrupt routine

Interrupt routines are special -> subroutines which can be started by events (external signals) in the machining process. A parts program block which is currently being worked through is interrupted and the position of the axes at the point of interruption is automatically saved.

Interrupts

All alarms and -> messages are output on the operator panel in plain text with the date and time and a symbol indicating the cancel criterion. The display is divided into alarms and messages.

1. Alarms and messages in the parts program
   Alarms and messages can be displayed as plain text directly from the parts program.
2. Alarms and messages from the PLC
   Alarms and messages from the machines can be displayed as plain text from the PLC program. No additional function block packages are required to do this.

Inverse time feedrate

With the SINUMERIK FM-NC and 840D, it is possible to program the time required to traverse the path of a block instead of programming the feedrate for the axis movement (G93).

J

Jog

Control operating mode (setup mode): In JOG mode, it is possible to set up the machine. Individual axes and spindles can be moved in this mode using the direction keys. Other functions available in JOG mode are -> reference point approach, -> repositioning and -> preset (setting an actual value).

K

Keyswitch

1. **S7-300**: The keyswitch is the operating mode switch of the -> CPU. A removable key is used to operate the keyswitch.
2. **840D/FM-NC**: The keyswitch on the -> machine control panel has 4 settings, to which functions are assigned by the operating system of the control. Three differently colored keys also belong to the keyswitch. These keys can be pulled out of the keyswitch in the indicated positions.

$K_{ü}$

Speed ratio

K_v

Servo gain factor, a control variable in a control loop.

L

Languages

The operator guidance display texts and the system messages are available in five system languages (diskette):
German, English, French, Italian, and **Spanish**
Two of the above languages are available and selectable in the control.

Leadscrew error compensation

Compensation for the mechanical inaccuracies of a leadscrew participating in the feed. The control uses stored deviation values for the compensation.

Limit speed

Maximum/minimum (spindle) speed: The maximum speed of a spindle may be limited by values defined in the machine data, the -> PLC or -> setting data.

Linear axis

The linear axis is an axis which, in contrast to a rotary axis, describes a straight line.

Linear interpolation

The tool travels along a straight line to the destination point while machining the workpiece.

Load memory

For the CPU 314 of the -> PLC, the load memory is equal to the -> working memory.

Look ahead

With the **look ahead** function, a configurable number of traversing blocks is read in advance in order to calculate the optimum machining velocity.

Look-ahead detection of contour violations

The control can recognize and signal the following types of collision:
1. The path distance is shorter than the tool radius.
2. The width of the inner corner is smaller than the tool diameter.

M

Machine Control operating area

Machine axes Axes which exist physically on the machine tool.

Machine control panel An operator panel on a machine tool with operating elements such as keys, rotary switches etc. and simple indicators such as LEDs. It is used to control the machine tool directly via the PLC.

Machine coordinate system System of coordinates based on the axes of the machine tool.

Machine zero A fixed point on the machine tool, which can be referenced by all (derived) measuring systems.

Machining channel Via a channel structure, parallel sequences of movements, such as positioning a loading gantry during machining, can shorten unproductive times. Here, a CNC channel must be regarded as a separate CNC control system with decoding, block preparation and interpolation.

Macro techniques Grouping of a set of instructions under a single identifier. The identifier in the program refers to the collected sequence of instructions.

Main block A block prefixed by ":" containing all the parameters required to start execution of a -> parts program.

Main program Parts program identified by a number or identifier in which further main programs, subroutines or -> cycles may be called.

Mains The term "network" describes the connection of several S7-300 and other terminal devices, e.g. a programming device, via -> interconnecting cables. A data exchange takes place over the network between the connected devices.

MDA Control operating mode: Manual Data Automatic. In MDA mode, it is possible to enter individual program blocks or sequences of blocks without reference to a main program or subroutine and to then execute them immediately via the NC start key.

Measuring circuit

- SINUMERIK FM-NC: The requisite control circuits for axes and spindles are integrated in the control module as standard. In total. a maximum of 4 axes and spindles can be realized, with a maximum of up to 2 spindles.
- SINUMERIK 840D: The signals from the sensors are analyzed in the SIMODRIVE 611D drive modules. The maximum configuration is 8 axes and spindles in total, with up to 5 spindles.

Messages
All messages programmed in the parts program and -> alarms recognized by the system are output on the operator panel in plain text with the date and time and a symbol indicating the cancel criterion. The display is divided into alarms and messages.

Metric measurement system
Standardized system of units: for lengths in millimeters (mm), meters (m), etc.

Mirroring
Mirroring inverts the signs of the coordinate values of a contour with respect to an axis. It is possible to mirror in relation to more than one axis at a time.

Mode group
At all times all of the axles/spindles are assigned to precisely one channel. Each channel is assigned to one operating mode group. The same -> mode is always assigned to the channels in a mode group.

Mounting rail
A mounting rail is used to attach the modules of an S7-300.

Multipoint interface
The multipoint interface (MPI) is a 9-pole Sub-D interface. A configurable number of devices can be connected to a multipoint interface and then communicate with each other.

- Programming devices
- Operator control and monitoring equipment
- Further automation systems

The parameter block "Multipoint Interface MPI" of the CPU contains the -> parameters which define the properties of the multipoint interface.

N

Names
Words in compliance with DIN 66025 are supplemented by identifiers (names) for variables (arithmetic variables, system variables, user variables), for subroutines, for keywords and for words with several address letters. In terms of the block structure, these supplements have the same significance as the words. Identifiers must be unique. The same identifier must not be used for different objects.

NC
Numerical Control: It incorporates all the components of the machine tool control system: -> NCK, -> PLC, -> MMC, -> COM.
Note: CNC (Computerized Numerical Control) is a more accurate term for the controls SINUMERIK 840D and FM-NC. MARS and Merkur controls.

NCK Numerical Control Kernel: Component of the NC control which executes -> parts programs and essentially coordinates the movements on the machine tool.

Node number The node number represents the "contact address" of a -> CPU or the -> programming device or any other intelligent periphery module if these are communicating via a -> network with each other. The node number is assigned to the CPU or the programming device with the S7 tool -> "S7 configuration".

NRK Numeric Robotic Kernel (operating system of the -> NCK)

NURBS Internal motion control and path interpolation are performed using NURBS (non-uniform rational B-splines). This provides a uniform internal method for all interpolations in the control (SINUMERIK 840D).

O

OEM For machine manufacturers who manufacture their own user interface or wish to integrate their own technology-specific functions in the control, free space has been left for individual solutions (OEM applications) for SINUMERIK 840D.

Offset memory Data range in the control in which the tool offset data are stored.

Operating mode An operating concept on a SINUMERIK control. The operating modes -> Jog, -> MDA and -> Automatic are defined.

Operator interface The user interface (UI) is the display medium for a CNC control in the form of a screen. It is laid out with eight horizontal and eight vertical softkeys.

Oriented spindle stop Stops the workpiece spindle with a specified orientation angle, e.g. to perform an additional machining operation at a specific position.

Oriented tool retraction RETTOOL: If machining is interrupted (because of tool breakage, for example), a program command can be used retract the tool with a defined orientation by a defined path.

Overall reset

During a general reset the following memories of the -> CPU are deleted:

- the -> working memory
- the read/write area of the -> load memory
- the -> system memory
- the -> backup memory

Override

Manual or programmable control feature which enables the user to override programmed feedrates or speeds in order to adapt them to a specific workpiece or material.

P

Parameters

S7-300: A distinction is made between 2 types of parameters:

1. Parameters of a STEP 7 instruction
   A parameter of a STEP 7 instruction is the address of the operand or a constant to be processed.
2. Parameter of a -> Parameterblock
   A parameter of a parameter block determines the behavior of a module.

Part program

A sequence of instructions to the NC control which combine to produce a specific -> workpiece by performing certain machining operation on a given -> blank Likewise, performing a certain machining operation on a specific -> blank.

Parts program management

Part programs can be organized according to -> workpieces. The size of the user memory determines the number of programs and the amount of data that can be managed. Each file (programs and data) can be assigned a name comprising up to 24 alphanumeric characters.

Path axis

Path axes are all the machining axes in the -> channel which are controlled by the -> interpolator such that they start, accelerate, stop and reach their end positions simultaneously.

Path feed

Path feed acts on -> path axes. It represents the geometrical sum of the feeds on the participating -> geometry axes.

Path velocity

The maximum programmable path velocity depends on the input resolution. For example, with a resolution of 0.1 mm the maximum programmable path velocity is 1000 m/min.

PG

Programming device

PLC Programmable Logic Control: -> Programmable Logic Controller. Component of the -> NC control: Programmable controller for processing the control logic of the machine tool.

PLC -> Programmable Logic Controller

PLC program memory SINUMERIK 840D:
PLC user memory is used to store the PLC user program and the user data together with the PLC basic program. The PLC user memory can be upgraded to 96 kByte with memory expansions.

PLC programming The PLC is programmed using the **STEP 7** software. The STEP 7 programming software is based on the **WINDOWS** operating system, and combines the STEP 5 programming functions with additional innovative functional developments.

Polar coordinates A coordinate system, which defines the position of a point on a plane in terms of its distance from the zero point and the angle formed by the radius vector with a defined axis.

Polynomial interpolation With polynomial interpolation, it is possible to generate many different curve characteristics, such as **straight line-, parabolic**-, and **exponential functions** (SINUMERIK 840D).

Position axis Axis which performs an auxiliary movement on a machine tool (e.g. tool magazine, pallet transport). Positioning axes are axes that do not interpolate with -> path axes.

Power On Switching the control off and back on again.

Preset The preset function can be used to redefine the control zero in the machine coordinate system. Preset does not move the axes, but a new position value is entered for the current axis positions.

Program
1. Control operating area
2. Sequence of instructions to the control.

Program block Program blocks contain the main programs and subprograms for the -> parts programs.

Programmable frames Programmable -> frames can be used to define new coordinate system starting points dynamically while the parts program is running. A distinction is made between absolute definition using a new frame and additive definition with reference to an existing starting point.

Programmable logic controller

Programmable logic controllers (PLC) are electronic controls, the function of which is stored as a program in the control unit. This means that the layout and wiring of the device do not depend on the function of the control. The programmable logic controller has the same structure as a computer; it consists of a CPU (central module) with memory, input/output modules and an internal bus system. The peripherals and the programming language are matched to the requirements of the control technology.

Programmable working area limitation

Limitation of the motion space of the tool to a space defined by programmed limitations.

Programming key

Characters and character sequences, which have a defined meaning in the programming language for -> parts programs (see programming guide).

Protection zone

Three-dimensional space within the -> working area which the tool tip is not permitted to enter.

Q

Quadrant error compensation

Contour errors at quadrant transitions, which arise as a result of changing friction conditions on the guideways, can be virtually entirely eliminated with the quadrant error compensation. Parameterization of the quadrant error compensation is performed by means of a circuit test.

R

R parameters

Arithmetic parameter for which the programmer of the -> parts program can freely assign or request values in the program.

Rapid traverse

The highest speed of an axis. It is used for example to move the tool from rest position to the -> workpiece contour or retract the tool from the contour.

Reference point approach

If the utilized distance measuring system is not an absolute value encoder then it is necessary to perform a reference point approach to ensure that the actual values returned by the measuring system match the machine coordinate values.

REPOS

1. Repositioning on the contour via the controls
   With the Repos function it is possible to re-approach the interruption point by means of the direction keys.

2. Repositioning on the contour via the program
   Several approach strategies are available through program commands: approaching the interruption point, approaching the block starting point, approaching the block end point, approaching a point on the path between the start of the block and the interruption point.

Retentive memory

Data areas in data blocks and times, counters and markers are retentive (non-volatile) if their contents are not lost when the system is restarted or the mains supply is disconnected.

Rigid tapping

This function allows threads to be tapped without a compensating chuck. By using the method whereby the spindle, as a rotary axis, and the drilling axis interpolate, threads can be cut to a precise final drilling depth (e.g. for blind hole threads) (requirement: spindles in axis operation).

Rotary axis

Rotary axes rotate a workpiece or tool to a defined angular position.

Rotary axis, continuously turning

Depending on the application, the traversing range of a rotary axis can be selected to be limited to less than 360 degrees or to be endlessly turning in both directions.
Endlessly-turning rotary axes are used for non-circular turning, grinding, and winding.

Rotation

Component of a -> frame which defines a rotation of the coordinate system through a specific angle.

Rounding axis

Rounding axes rotate a workpiece or tool to an angular position corresponding to an indexing grid. When a grid index is reached, the rounding axis is "in position".

S

S7 configuration

S7 configuration is a tool with the aid of which modules can be parameterized. With S7 configuration, various -> parameter blocks of the -> CPU and the I/O modules are set on the -> programming unit. These parameters are transmitted to the CPU.

S7-300 bus

The S7-300 bus is a serial databus by means of which the modules communicate with each other and are supplied with the required voltage. The connections between the modules are made with the -> bus connectors.

Safety functions

The controls contain watchdog monitors which are always active. These monitors detect problems in the CNC, PLC or machine in time to prevent damage to workpiece, tool or machine as far as possible. In the event of a malfunction the machining sequence is interrupted and the drives are stopped, the cause of the malfunction is saved and displayed as an alarm. At the same time, the PLC is informed that a CNC alarm is pending.

Scaling

Component of a -> frame, which causes axis-specific scale modifications.

Serial RS-232-C interface

For data input/output there is

- a serial RS-232-C interface on the MMC module MMC100, and on
- MMC modules PCU 50/70 two RS-232-C interfaces

are available. Machining programs and manufacturer and user data can be loaded and saved via these interfaces.

Services

Control operating area

Set

A section of a -> parts program terminated with a line feed. A distinction is made between -> main blocks and -> subblocks.

Setting data

Data which communicate properties of the machine tool to the NC control in a way defined by the system software.

SIMODRIVE 611

- SINUMERIK FM-NC offers an analog ±10V interface to the SIMODRIVE 611A converter system.
- The SINUMERIK 840D control system is connected to the SIMODRIVE 611D converter system by means of a high-speed digital parallel bus.

Simulator module

A simulator module is a module

- on which digital input variables can be simulated via control elements and
- digital output variables can be displayed.

Softkey

A key whose name appears on an area of the screen. The selection of keys displayed is adapted dynamically to the operating situation. The freely assignable function keys (softkeys) are assigned defined functions in the software.

Software limit switches

Software limit switches limit the traversing range of an axis and prevent the slide from striking the hardware limit switch. 2 pairs of values can be assigned on each axis, which can then be activated separately via the -> PLC.

Spindles

The spindle functionality is divided into two power ranges.

1. Spindles: Speed or position-controlled analog spindle drives
   ±10V (SINUMERIK FM-NC)
   digital (SINUMERIK 840D)
2. Auxiliary spindles: speed-controlled spindle drives, "auxiliary spindle" function package e.g. for driven tools.

Spline interpolation

Using spline interpolation, the control can obtain a very smooth curve from just a few defined interpolation points along a set contour.

Standard cycles

Standard cycles are available for frequently recurring machining tasks.

- for drilling/milling technology
- for turning technology

In the "Program" operating area, the available cycles are listed under the menu "Cycle Support". After selecting the desired machining cycle the required parameters for the value assignment are displayed in clear text.

Structure

- The SINUMERIK FM-NC is integrated into the CPU row of the SIMATIC S7-300. The module is 200 mm wide and fully encapsulated, with the same external structure as the SIMATIC S7-300 modules.
- The SINUMERIK 840D is integrated as a compact module into the SIMODRIVE 611D converter system. The dimensions correspond to a 50 mm wide SIMODRIVE 611D module. The SINUMERIK 840D module comprises the NCU module and the NCU box.

Subblock

Block prefixed by "N" containing information for a machining step such as position data.

Subroutine

A sequence of instructions in a -> parts program, which can be called repeatedly with various defining parameters. The subroutine is called from a main program. It is not possible to block every subroutine against unauthorized reading and displaying. -> Cycles are a form of subroutine.

Synchronization

Instructions in -> parts programs for coordination of sequences in different -> channels at specific machining points.

Synchronized actions

1. Auxiliary function output
   During the workpiece machining, technology functions (-> auxiliary functions) can be issued from the CNC program to the PLC. These auxiliary functions are used for example to control additional equipment for the machine tool, such as quills, grabbers, clamping chucks etc.
2. Fast auxiliary function output
   For switching functions which are time-critical, the confirmation times for the -> auxiliary functions are minimized, and unnecessary stopping points in the machining process can be avoided.

Synchronized axes

Synchronized axes take the same time to traverse as the geometry axes take for their path.

System memory

The system memory is a memory in the CPU in which the following data are stored:

- Data required by the operating system
- The operands times, counters, markers

System variable

A variable, which exists although it has not been programmed by the -> parts program programmer. It is defined by a data type and the variable name preceded by the character $.
See also -> user-defined variable.

T

Teach In

Teach In can be used to create or correct parts programs. The individual program blocks can be input via the keyboard and traversed immediately. Positions approached via the direction keys or the handwheel can also be saved. Additional details such as G functions, feeds or M functions can be input in the same block.

Text editor

-> Editor

Tool

A tool is a software tool for inputting and changing the -> parameters of a parameter block. Tools include:

- S7-Configuration
- S7-TOP
- S7-Info

Tool offset

By programming a **T function** (5 integer decades) in the block, you can select the tool. Up to 9 cutting edges (D addresses) can be assigned to every T number. The number of tools to be managed in the controller is specified in the configuring data.

Tool radius compensation
In order to be able to program a desired -> workpiece contour directly, the control must traverse a path equidistant to the programmed contour, taking into account the radius of the tool used. (G41/G42).

Tools
A part used on the machine tool for machining. Examples of tools include lathe tools, milling cutters, drills, laser beams, etc.

Transformation
Programming in a Cartesian coordinate system, execution in a non-Cartesian coordinate system (e.g. with machine axes as rotary axes).

Traversing range
The maximum permissible traversing range on linear axes is ± 9 decades. The absolute value depends on the selected input sensitivity and positioning resolution and the system of units used (inches or metric).

U

User memory
All program and data, such as part programs, subroutines, comments, tool compensations, and work offsets/frames, as well as channel- and program user data can be stored in the shared CNC user memory.

User program
User programs for the S7-300 automation systems are created using the programming language STEP 7. The user program has a modular layout and consists of individual blocks.
The basic block types are:
Code blocks: these blocks contain the STEP 7 commands.
Data blocks: these blocks contain the constants and variables for the STEP 7 program.

User-defined variable
The user can declare user-defined variables for any use in the -> parts program or data block (global user data). A definition contains a data type specification and the variable name. See also -> system variable.

V

Variable definition
A variable definition includes the specification of a data type and a variable name. The variable names can be used to access the value of the variables.

Velocity control In order to be able to achieve an acceptable traversing velocity on very short traverse movements within a single block, predictive velocity control can be set over several blocks (-> look ahead).

Vocabulary words Words with a specific notation, which have a defined meaning in the programming language for -> parts programs.

W

Working area Three-dimensional zone into which the tool tip can be moved on account of the physical design of the machine tool.
See also -> protection zone.

Working area limitation With the aid of the working area limitation, the traversing range of the axes can be further restricted in addition to the limit switches. One value pair per axis may be used to describe the protected working area.

Working memory The working area is a RAM area in the -> CPU which is accessed by the processor to access the user program during program execution.

Workpiece Part to be created/machined by the machine tool. A workpiece is saved as a separate program in a directory.

Workpiece contour Setpoint contour of the -> workpiece to be created/machined.

Workpiece coordinate system The starting position of the workpiece coordinate system is the -> workpiece zero. In machining operations programmed in the workpiece coordinate system, the dimensions and directions refer to this system.

Workpiece zero The workpiece zero is the starting point for the -> workpiece coordinate system. It is defined in terms of the distance from the machine zero.

X

Y

Z

Zero offset

Specification of a new reference point for a coordinate system through reference to an existing zero point and a -> frame.

1. Adjustable
   SINUMERIK FM-NC: Four independent zero offsets can be selected for each CNC axis.
   SINUMERIK 840D: A configurable number of adjustable zero offsets is available for each CNC axis. The offsets which can be selected via G functions are effective on an alternating basis.
2. External
   In addition to all displacements which define the position of the workpiece zero point, an external zero offset can be superimposed via
   - a handwheel (DRF offset) or
   - the PLC.
3. Programmable
   Zero offsets are programmable for all path and positioning axes with the TRANS command.

C References

General Documentation

/BU/ SINUMERIK & SIMODRIVE, Automation Systems for Machine Tools
Catalog NC 60
Order No.: E86060-K4460-A101-A9-7600

/IKPI/ Industrial Communications and Field Devices
Catalog IK PI
Order No.: E86060-K6710-A101-B2-7600

/ST7/ SIMATIC
Products for Totally Integrated Automation and Micro Automation
Catalog ST 70
Order No.: E86060-K4670-A111-A8-7600

/Z/ MOTION-CONNECT
Connections & System Components for SIMATIC, SINUMERIK, MASTERDRIVES, and SIMOTION
Catalog NC Z
Order No.: E86060-K4490-A001-B1-7600

Safety Integrated
Application Manual
The safety program for industries of the world
Order No.: 6ZB5000-0AA02-0BA0

Electronic Documentation

/CD1/ The SINUMERIK System (03.04 Edition)
DOC ON CD
(includes all SINUMERIK 840D/840Di/810D/802- and SIMODRIVE publications)
Order No.: 6FC5298-7CA00-0BG0

User Documentation

/AUK/ SINUMERIK 840D/810D (09.99 Edition)
Short Guide **AutoTurn**
Order No.: 6FC5298-4AA30-0BP2

/AUP/ SINUMERIK 840D/810D (02.02 Edition)
Operator's Guide **AutoTurn Graphic Programming System**
Programming/Setup
Order No.: 6FC5298-4AA40-0BP3

/BA/ SINUMERIK 840D/810D (10.00 Edition)
Operator's Guide **MMC**
Order No.: 6FC5298-6AA00-0BP0

/BAD/ SINUMERIK 840D/840Di/810D (03.04 Edition)
Operator's Guide **HMI Advanced**
Order No.: 6FC5298-6AF00-0BP3

/BAH/ SINUMERIK 840D/840Di/810D (03.04 Edition)
Operator's Guide **HT 6**
Order No.: 6FC5298-0AD60-0BP3

/BAK/ SINUMERIK 840D/840Di/810D (02.01 Edition)
Short Guide Operation
Order No.: 6FC5298-6AA10-0BP0

/BAM/ SINUMERIK 810D/840D (08.02 Edition)
Operation/Programming **ManualTurn**
Order No.: 6FC5298-6AD00-0BP0

/BAS/ SINUMERIK 840D/840Di/810D (11.03 Edition)
Operation/Programming **ShopMill**
Order No.: 6FC5298-6AD10-0BP2

/BAT/ SINUMERIK 840D/810D (06.03 Edition)
Operation/Programming **ShopTurn**
Order No.: 6FC5298-6AD50-0BP2

/BEM/ SINUMERIK 840D/810D (03.04 Edition)
Operator's Guide **HMI Embedded**
Order No.: 6FC5298-6AC00-0BP3

/BNM/ SINUMERIK 840D/840Di/810D (03.04 Edition)
User's Guide **Measuring Cycles**
Order No.: 6FC5298-7AA70-0BP3

/BTDI/ SINUMERIK 840D/840Di/810D (04.03 Edition)
Motion Control Information System (MCIS)
User's Guide **Tool Data Information**
Order No.: 6FC5297-6AE01-0BP0

/CAD/ SINUMERIK 840D/840Di/810D (03.02 Edition)
Operator's Guide **CAD Reader**
Order No.: (included in online help)

/DA/	SINUMERIK 840D/840Di/810D **Diagnostics Guide** Order No.: 6FC5298-7AA20-0BP0	(03.04 Edition)
/KAM/	SINUMERIK 840D/810D Short Guide **ManualTurn** Order No.: 6FC5298-5AD40-0BP0	(04.01 Edition)
/KAS/	SINUMERIK 840D/810D Short Guide **ShopMill** Order No.: 6FC5298-5AD30-0BP0	(04.01 Edition)
/KAT/	SINUMERIK 840D/810D Short Guide **ShopTurn** Order No.: 6FC5298-6AF20-0BP0	(07.01 Edition)
/PG/	SINUMERIK 840D/840Di/810D Programming Guide **Fundamentals** Order No.: 6FC5298-7AB00-0BP0	(03.04 Edition)
/PGA/PGA1	SINUMERIK 840D/840Di/810D Programming Guide **Advanced / System Variables** Order No.: 6FC5298-7AB10-0BP0 / 6FC5298-7AE10-0BP0	(03.04 Edition)
/PGK/	SINUMERIK 840D/840Di/810D Short Guide **Programming** Order No.: 6FC5298-7AB30-0BP0	(03.04 Edition)
/PGM/	SINUMERIK 840D/840Di/810D Programming Guide **ISO Milling** Order No.: 6FC5298-6AC20-0BP2	(11.02 Edition)
/PGT/	SINUMERIK 840D/840Di/810D Programming Guide **ISO Turning** Order No.: 6FC5298-6AC10-0BP2	(11.02 Edition)
/PGZ/	SINUMERIK 840D/840Di/810D Programming Guide **Cycles** Order No.: 6FC5 298-7AB40-0BP0	(03.04 Edition)
/PI/	PCIN 4.4 Software for Data Transfer to/from **MMC Modules** Order number: 6FX2060-4AA00-4XB0 (English, German, French) Order from: WK Fürth	
/SYI/	SINUMERIK 840Di **System Overview** Order No.: 6FC5298-6AE40-0BP0	(02.01 Edition)

Manufacturer/Service Documentation

a) Lists

/LIS/ SINUMERIK 840D/840Di/810D (03.04 Edition)
SIMODRIVE 611D
Lists
Order No.: 6FC5297-7AB70-0BP0

b) Hardware

/ASAL/ SIMODRIVE 611, MASTERDRIVES VC/MC (10.03 Edition)
Planning Guide General Information for **Asynchronous Motors**
Order No.: 6SN1197-0AC62-0BP0

/APH2/ SIMODRIVE 611 (10.03 Edition)
Planning Guide **Asynchronous Motors 1PH2**
Order No.: 6SN1197-0AC63-0BP0

/APH4/ SIMODRIVE 611 (10.03 Edition)
Planning Guide **Asynchronous Motors 1PH4**
Order No.: 6SN1197-0AC64-0BP0

/APH7S/ SIMODRIVE 611 (03.04 Edition)
Planning Guide **Asynchronous Motors 1PH7**
Order No.: 6SN1197-0AC66-0BP0

/APH7M/ MASTERDRIVES MC (04.04 Edition)
Planning Guide **Asynchronous Motors 1PH7**
Order No.: 6SN1197-0AC65-0BP0

/APL6/ MASTERDRIVES VC/MC (03.04 Edition)
Planning Guide **Asynchronous Motors 1PL6**
Order No.: 6SN1197-0AC67-0BP0

/BH/ SINUMERIK 840D/840Di//810D (11.03 Edition)
Operator Components Manual (HW)
Order No.: 6FC5297-6AA50-0BP3

/BHA/ SIMODRIVE Sensor (03.03 Edition)
User Guide (HW) **Absolute Position Sensor with Profibus DP**
Order No.: 6SN1197-0AB10-0YP2

/EMV/ SINUMERIK, SIROTEC, SIMODRIVE, SIMOTION (06.99 Edition)
Planning Guide **EMC Installation Guide**
Order No.: 6FC5297-0AD30-0BP1

The current Declaration of Conformity is available under the following Internet address:
http://www4.ad.siemens.de

Please enter the ID no. 15257461 in the "Search" field (top right) and click on "go".

/GHA/ SINUMERIK/SIMOTION (02.03 Edition)
ADI4 - Analog Drive Interface for 4 Axes
Manual
Order No.: 6FC5297-0BA01-0BP1

/PFK6/ SIMODRIVE 611, MASTERDRIVES MC (05.03 Edition)
Planning Guide **1FK6 Three-Phase AC Servomotors**
Order No.: 6SN1197-0AD05-0BP0

/PFK7/ SIMODRIVE 611, MASTERDRIVES MC (01.03 Edition)
Planning Guide **1FK7 Three-Phase AC Servomotors**
Order No.: 6SN1197-0AD06-0BP0

/PFS6/ MASTERDRIVES MC (07.03 Edition)
Planning Guide **1FS6 Three-Phase AC Servomotors**
Order No.: 6SN1197-0AD08-0BP0

/PFT5/ SIMODRIVE 611 (05.03 Edition)
Planning Guide **1FT5 Three-Phase AC Servomotors**
Order No.: 6SN1197-0AD01-0BP0

/PFT6/ SIMODRIVE 611, MASTERDRIVES MC (01.04 Edition)
Planning Guide **1FT6 Three-Phase AC Servomotors**
Order No.: 6SN1197-0AD02-0BP0

/PFU/ SINAMICS, MASTERDRIVES, (09.03 Edition)
MICROMASTER
SIEMOSYN Motors 1FU8
Order No.: 6SN1197-0AC80-0BP0

/PHC/ SINUMERIK 810D (11.02 Edition)
Configuring Manual CCU **(HW)**
Order No.: 6FC5297-6AD10-0BP1

/PHD/ SINUMERIK 840D (11.03 Edition)
Configuring Manual **NCU (HW)**
Order No.: 6FC5297-6AC10-0BP3

/PJAL/ SIMODRIVE 611, MASTERDRIVES MC (01.04 Edition)
Planning Guide **Three-Phase Servomotors**
General Part for 1FT / 1FK Motors
Order No.: 6SN1197-0AD07-0BP1

/PJAS/ SIMODRIVE 611, MASTERDRIVES VC/MC (06.04 Edition)
Planning Guide Asynchronous Motors
Contents: **General Part, 1PH2, 1PH4, 1PH7, 1PL6**
Order No.: 6SN1197-0AC61-0BP0

/PJFE/ SIMODRIVE (03.04 Edition)
Planning Guide **1FE1 Built-In Synchronous Motors**
Three-Phase AC Motors for Main Spindle Drives
Order No.: 6SN1197-0AC00-0BP5

/PJF1/ SIMODRIVE (12.02 Edition)
Installation Guide **1FE1 051.-1FE1 147**. **Built-In Synchronous Motors**
AC Motors for Main Spindle Drives
Order No.: 610.43000.02

/PJLM/ SIMODRIVE (06.02 Edition)
Planning Guide **1FN1, 1FN3 Linear Motors**
ALL General Information about Linear Motor
1FN1 1FN1 Three-Phase AC Linear Motor
1FN3 1FN3 Three-Phase AC Linear Motor
CON Connections
Order No.: 6SN1197-0AB70-0BP4

/PJM2/ SIMODRIVE 611, MASTERDRIVES MC (03.04 Edition)
Planning Guide **Servomotors**
Contents: General Part, 1FT5, 1FT6, 1FK6, 1FK7, 1FS6
Order No.: 6SN1197-0AC20-0BP0

/PJTM/ SIMODRIVE (05.03 Edition)
Planning Guide **1FW6 Built-In Torque Motors 1FW6**
Order No.: 6SN1197-0AD00-0BP1

/PJU/ SIMODRIVE 611 (02.03 Edition)
Planning Guide **Converters**
Order No.: 6SN1197-0AA00-0BP6

/PKTM/ MASTERDRIVES (03.04 Edition)
Planning Guide **Torque Motors 1FW3**
Order No.: 6SN1197-0AC70-0BP0

/PMH/ SIMODRIVE Sensor (07.02 Edition)
Configuring/Installation Guide
Hollow-Shaft Measuring System SIMAG H
Order No.: 6SN1197-0AB30-0BP1

/PMH2/ SIMODRIVE Sensor (03.04 Edition)
Configuring/Installation Guide
Hollow-Shaft Measuring System SIMAG H2
Order No.: 6SN1197-0AB31-0BP0

/PMHS/ SIMODRIVE (12.00 Edition)
Installation Guide **Measuring System for Main Spindle Drives**
SIZAG2 Toothed-Wheel Encoder
Order No.: 6SN1197-0AB00-0YP3

/PMS/ SIMODRIVE (03.04 Edition)
Planning Guide
ECO Motor Spindle for Main Spindle Drives
Order No.: 6SN1197-0AD04-0BP1

/PPH/ SIMODRIVE (12.01 Edition)
Planning Guide **1PH2 / 1PH4 / 1PH7 Motors**
AC Induction Motors for Main Spindle Drives
Order No.: 6SN1197-0AC60-0BP0

/PPM/ SIMODRIVE (11.01 Edition)
Planning Guide
Hollow-Shaft Motors for **1PM4 and 1PM6** Main Spindle Drives
Order No.: 6SN1197-0AD03-0BP0

c) Software

/FB1/ SINUMERIK 840D/840Di/810D/FM-NC (03.04 Edition)
Description of Function **Basic Machine (Part 1)**
(the various sections are listed below)
Order No.: 6FC5297-7AC20-0BP0

- A2 Various Interface Signals
- A3 Axis Monitoring, Protection Zones
- B1 Continous-Path Mode, Exact Stop and Look Ahead
- B2 Acceleration
- D1 Diagnostic Tools
- D2 Interactive Programming
- F1 Traverse to Fixed Stop
- G2 Velocities, Setpoint/Actual Value Systems, Closed-Loop Control
- H2 Output of Auxiliary Functions to PLC
- K1 Mode Group, Channel, Program Operation Mode
- K2 Axes, Coordinate Systems, Frames, Actual-Value System for Workpiece, External Zero Offset
- K4 Communication
- N2 EMERGENCY STOP
- P1 Traverse Axes
- P3 Basic PLC Program
- R1 Reference Point Approach
- S1 Spindles
- V1 Feeds
- W1 Tool Offset

/FB2/ SINUMERIK 840D/840Di/810D (03.04 Edition)
Description of Functions **Extended Functions (Part 2)**
including FM-NC: Turning, Stepper Motor
(the various sections are listed below)
Order No.: 6FC5297-7AC30-0BP0

- A4 Digital and Analog NCK I/Os
- B3 Several Operator Panels and NCUs
- B4 Operation via PG/PC
- F3 Remote Diagnostics
- H1 JOG with/without Handwheel
- K3 Compensations
- K5 Mode Groups, Channels, Axis Replacement
- L1 FM NC Local Bus
- M1 Kinematic Transformation
- M5 Measurement
- N3 Software Cams, Position Switching Signals
- N4 Punching and Nibbling
- P2 Positioning Axes
- P5 Oscillation
- R2 Rotary Axes
- S3 Synchronous Spindle
- S5 Synchronized Actions (SW 3 and lower, higher /FBSY/)
- S6 Stepper Motor Control
- S7 Memory Configuration
- T1 Indexing Axes
- W3 Tool Change
- W4 Grinding

/FB3/ SINUMERIK 840D/840Di/810D (08.04 Edition)
Description of Functions **Special Functions (Part 3)**
(the various sections are listed below)
Order No.: 6FC5297-7AC80-0BP1
F2 3-Axis to 5-Axis Transformation
G1 Gantry Axes
G3 Clock Times
K6 Contour Tunnel Monitoring
M3 Coupled Motion and Leading Value Coupling
S8 Constant Workpiece Speed for Centerless Grinding
T3 Tangential Control
TE0 Installation and Activation of Compile Cycles
TE1 Clearance Control
TE2 Analog Axes
TE3 Master-Slave for Drives
TE4 Transformation Package Handling
TE5 Setpoint Exchange
TE6 MCS Coupling
TE7 Retrace Support
TE8 Pulse-Independent Path-Synchronized Switching Signal Output
V2 Preprocessing
W5 3D Tool Radius Compensation

/FBA/ SIMODRIVE 611D/SINUMERIK 840D/810D (03.04 Edition)
Description of Functions **Drive Functions**
(the various sections are listed below)
Order No.: 6SN1197-0AA80-1BP1
DB1 Operating Messages/Alarm Reactions
DD1 Diagnostic Functions
DD2 Speed Control Loop
DE1 Extended Drive Functions
DF1 Enable Commands
DG1 Encoder Parameterization
DL1 Linear Motor MD
DM1 Calculating Motor/Power Section Parameters and Controller Data
DS1 Current Control Loop
DÜ1 Monitors/Limitations

/FBAN/ SINUMERIK 840D/SIMODRIVE 611 DIGITAL (02.00 Edition)
Description of Functions **ANA MODULE**
Order No.: 6SN1197-0AB80-0BP0

/FBD/ SINUMERIK 840D (07.99 Edition)
Description of Functions **Digitizing**
Order No.: 6FC5297-4AC50-0BP0
DI1 Start-Up
DI2 Scan with Tactile Sensor (scancad scan)
DI3 Scan with Laser (scancad laser)
DI4 Milling Program Generation (scancad mill)

/FBDM/ SINUMERIK 840D/840Di/810D (09.03 Edition)
Description of Functions DNC **NC Program Management**
Order No.: 6FC5297-1AE81-0BP0

/FBDN/ SINUMERIK 840D/840Di/810D (03.03 Edition)
Motion Control Information System (MCIS)
Description of Functions **DNC NC Program Management**
Order No.: 6FC5297-1AE80-0BP0
DN1 DNC Plant / DNC Cell
DN2 DNC IFC SINUMERIK, NC Data Transfer via Network

/FBFA/ SINUMERIK 840D/840Di/810D (07.04 Edition)
Description of Functions **ISO Dialects for SINUMERIK**
Order No.: 6FC5297-6AE10-0BP4

/FBFE/ SINUMERIK 840D/810D (03.04 Edition)
Motion Control Information System (MCIS)
Description of Functions **Remote Diagnosis**
Order No.: 6FC5297-0AF00-0BP3
FE1 Remote Diagnosis ReachOut
FE3 Remote Diagnosis pcAnywhere

/FBH/ SINUMERIK 840D/840Di/810D (11.02 Edition)
HMI Configuring Package
Order No.: (supplied with the software)

Part 1 User's Guide
Part 2 Description of Functions

/FBH1/ SINUMERIK 840D/840Di/810D (03.03 Edition)
HMI Configuring Package
ProTool/Pro Option SINUMERIK
Order No.: (supplied with the software)

/FBHL/ SINUMERIK 840D/SIMODRIVE 611 digital (10.03 Edition)
Description of Functions **HLA Module**
Order No.: 6SN1197-0AB60-0BP3

/FBIC/ SINUMERIK 840D/840Di/810D (06.03 Edition)
Motion Control Information System (MCIS)
Description of Functions **TDI Ident Connection**
Order No.: 6FC5297-1AE60-0BP0

/FBMA/ SINUMERIK 840D/810D (08.02 Edition)
Description of Functions **ManualTurn**
Order No.: 6FC5297-6AD50-0BP0

/FBO/ SINUMERIK 840D/810D (09.01 Edition)
Description of Functions Configuring **OP 030 Operator Interface**
(the various sections are listed below)
Order No.: 6FC5297-6AC40-0BP0
BA Operator's Guide
EU Development Environment (Configuring Package)
PSE Introduction to Configuring of Operator Interface
IK Screen Kit: Software Update and Configuration

/FBP/ SINUMERIK 840D (03.96 Edition)
Description of Functions **C-PLC Programming**
Order No.: 6FC5297-3AB60-0BP0

/FBR/ SINUMERIK 840D/840Di/840D (01.04 Edition)
IT Solutions
Description of Functions **RPC SINUMERIK Computer Link**
Order No.: 6FC5297-6AD61-0BP0
NFL Host Computer Interface
NPL PLC/NCK Interface

/FBSI/ SINUMERIK 840D/SIMODRIVE (11.03 Edition)
Description of Functions SINUMERIK **Safety Integrated**
Order No.: 6FC5297-6AB80-0BP2

/FBSP SINUMERIK 840D/840Di/810D (11.03 Edition)
Description of Functions **ShopMill**
Order No.: 6FC5297-6AD80-0BP2

/FBST/ SIMATIC (01.01 Edition)
Description of Functions **FM STEPDRIVE/SIMOSTEP**
Order No.: 6SN1197-0AA70-0YP4

/FBSY/ SINUMERIK 840D/810D (03.04 Edition)
Description of Functions **Synchronized Actions**
Order No.: 6FC5297-7AD40-0BP2

/FBT/ SINUMERIK 840D/810D (03.04 Edition)
Description of Functions **ShopTurn**
Order No.: 6FC5297-6AD70-0BP2

/FBTC/ SINUMERIK 840D/810D (01.02 Edition)
IT Solutions
Description of Functions **Tool Data Communication SinTDC**
Order No.: 6FC5297-5AF30-0BP0

/FBTD/ SINUMERIK 840D/810D (02.01 Edition)
IT Solutions
Description of Functions **Tool Information System (SinTDI)**
with Online Help
Order No.: 6FC5297-6AE00-0BP0

/FBTP/ SINUMERIK 840D/840Di/810D (01.03 Edition)
Motion Control Information System (MCIS)
Description of Functions **TPM Total Productive Maintenance**
Order No.: Document is supplied with the software

/FBU/ SIMODRIVE 611 universal/universal E (07.04 Edition)
Description of Functions **Closed-Loop Control Component for Speed Control and Positioning**
Order No.: 6SN1197-1AB20-0BP9

/FBU2/ SIMODRIVE 611 **universal** (04.02 Edition)
Installation Guide
(enclosed with SIMODRIVE 611 universal)

/FBW/ SINUMERIK 840D/810D (11.02 Edition)
Description of Functions **Tool Management**
Order No.: 6FC5297-6AC60-0BP1

/HBA/ SINUMERIK 840D/840Di/810D (03.02 Edition)
Manual **@Event**
Order No.: 6AU1900-0CL20-0BA0

/HBI/ SINUMERIK 840Di (03.04 Edition)
Manual **SINUMERIK 840Di**
Order No.: 6FC5297-7AE60-0BP0

/INC/ SINUMERIK 840D/840Di/810D (06.03 Edition)
System Description **Commissioning Tool SINUMERIK SinuCOM NC**
Order No.: (an integral part of the online help for the start-up tool)

/PGA1/ SINUMERIK 840D/840Di/810D (03.04 Edition)
List Manual **System Variables**
Order No.: 6FC5297-6EA10-0BP0

/PJE/ SINUMERIK 840D/810D (08.01 Edition)
Description of Functions **Configuring Package HMI Embedded**
Software Update, Configuration, Installation
Order No.: 6FC5297-6EA10-0BP0

/PS/ SINUMERIK 840D/810D (09.03 Edition)
Planning Guide **Configuring Syntax**
(the document PS Configuring Syntax is supplied with the software and available as a pdf file)

/POS1/ SIMODRIVE **POSMO A** (08.03 Edition)
User Guide's **Distributed Positioning Motor on PROFIBUS DP**
Order No.: 6SN2197-0AA00-0BP6

/POS2/ SIMODRIVE **POSMO A** (08.03 Edition)
Installation Guide (enclosed with POSMO A)

/POS3/ SIMODRIVE POSMO SI/CD/CA (07.04 Edition)
User Guide's **Distributed Servo Drive Systems**
Order No.: 6SN2197-0AA20-0BP7

/POS4/ SIMODRIVE **POSMO SI** (08.03 Edition)
Installation Guide (enclosed with POSMO SI)

/POS5/ SIMODRIVE **POSMO CD/CA** (08.03 Edition)
Installation Guide (enclosed with POSMO CD/CA)

/S7H/ SIMATIC S7-300 (2002 Edition)
Installation Manual **Technological Functions**
Order No.: 6ES7398-8AA03-8BA0
- Reference Manual: CPU Data (HW Description)
- Reference Manual: Module Data

/S7HT/	SIMATIC S7-300 Manual **STEP 7, Fundamentals, V. 3.1** Order No.: 6ES7810-4CA02-8BA0	(03.97 Edition)
/S7HR/	SIMATIC S7-300 Manual **STEP 7, Reference Manuals, V. 3.1** Order No.: 6ES7810-4CA02-8BR0	(03.97 Edition)
/S7S/	SIMATIC S7-300 **FM 353** Positioning Module **for Stepper Drive** Order together with configuring package	(04.02 Edition)
/S7L/	SIMATIC S7-300 **FM 354** Positioning Module **for Servo Drive** Order together with configuring package	(04.02 Edition)
/S7M/	SIMATIC S7-300 **FM 357.2 Multimodule** for Servo and Stepper Drives Order together with configuring package	(01.03 Edition)
/SP/	SIMODRIVE 611-A/611-D **SimoPro 3.1** Program for Configuring Machine-Tool Drives Order number: 6SC6111-6PC00-0BA❑ Order from: WK Fürth	

d) Installation and Start-up

/BS/ SIMODRIVE 611 analog (10.00 Edition)
Description **Start-Up Software for Main Spindle and Asynchronous Motor Modules Version 3.20**
Order No.: 6SN1197-0AA30-0BP1

/IAA/ SIMODRIVE 611A (10.00 Edition)
Installation and Start-Up Guide
Order No.: 6SN1197-0AA60-0BP6

/IAC/ SINUMERIK 810D (11.02 Edition)
Installation and Start-Up Guide
(including description of SIMODRIVE 611D start-up software)
Order No.: 6FC5297-6AD20-0BP1

/IAD/ SINUMERIK 840D/SIMODRIVE 611D (03.04 Edition)
Installation and Start-Up Guide
(including description of SIMODRIVE 611D start-up software)
Order No.: 6FC5297-7AB10-0BP0

/IAM/ SINUMERIK 840D/840Di/810D (03.04 Edition)
Installation and Start-Up Guide **HMI/MMC**
Order No.: 6FC5297-6AE20-0BP4
AE1 Updates/Supplements
BE1 Expanding the Operator Interface
HE1 Online Help
IM2 Start-Up HMI Embedded
IM4 Start-Up HMI Advanced
TX1 Creating Foreign Language Texts

■

D Index

To
Siemens AG

A&D MC BMS
P.O. Box 3180

D-91050 Erlangen, Germany

Phone ++49-(0)180-5050-222 [Hotline]

Fax ++49-(0)9131-98-2176 [Documentation]

E-mail: motioncontrol.docu@siemens.com

Suggestions

Corrections

For publication:

SINUMERIK 840D/810D

HMI Embedded

User Documentation

From

Name:

Company/Dept.

Address: _____

Zip code: _____ City: _____

Phone: _____ / _____

Fax: _____ / _____

Operator's Guide

Order no.: 6FC5298-6AC00-0BP3
03.04 Edition

Should you come across any printing errors when reading this publication, please notify us on this sheet. Suggestions for improvements are also welcome.

Suggestions and/or corrections

Overview of SINUMERIK 840D/840Di/810D Documentation (04.2004)

General Documentation

- *SINUMERIK* 840D/810D — Brochure
- *SINUMERIK* 840D/840Di/810D/ — Catalog Ordering Info. NC 60 ***)**
- *Safety Integrated* Application Manual — **Safety Integrated** Application Manual

User Documentation

- *SINUMERIK* 840D/810D/FM-NC — **AutoTurn**
  - Short Guide
  - Programming/ Setup
- *SINUMERIK* 840D/840Di/810D — Operator's Guide **– HT 6**
- *SINUMERIK* 840D/840Di/810D — **Diagnostics Guide *)**
- *SINUMERIK* 840D/840Di/810D — **Operator's Guide *)**
  - Short Guide
  - HMI Embedded
  - HMI Advanced

User Documentation

- *SINUMERIK* 840D/840Di/810D — **Programming Guide**
  - Short Guide
  - Fundamentals ***)**
  - Advanced ***)**
  - Cycles
  - Measuring Cycles
  - ISO Turning/Milling
  - Lists **System Variables**
- *SINUMERIK* 840D/810D — Operator's Guide
  - **ManualTurn**
  - Short Guide ManualTurn
  - **ShopMill**
  - Short Guide ShopMill
  - **ShopTurn**
  - Short Guide ShopTurn
- *SINUMERIK* 840Di — **System Overview**

Manufacturer/Service Documentation

- *SINUMERIK* — **Configuring** (HW) ***)**
  - 810D
  - 840D
- *SINUMERIK* 840D/840Di/810D — **Operator Components** (HW) ***)**
- *SINUMERIK* 840D/840Di/810D — Description of Functions
  - **ManualTurn**
  - **ShopMill**
  - **ShopTurn**

Manufacturer/Service Documentation

- *SINUMERIK* 611D 840D/810D — Description of Functions **Synchronized Actions**
- *SINUMERIK SIMODRIVE* 840D/840Di/810D — Description of Functions **Drive Functions *)**
- *SINUMERIK* 840D/840Di/810D — **Description of Functions**
  - Basic Machine ***)**
  - Extended Functions
  - Special Functions
- *SINUMERIK* 840D/810D — Description of Functions **Tool Management**
- *SINUMERIK* 840D/810D — **Configuring Kit** HMI Embedded
- *SINUMERIK* 840D/810D — Description of Functions Operator Interface **OP 030**
- *SINUMERIK* 840D/810D — **MCIS**
  - Computer Link
  - Tool Data Inform. Syst.
  - NC Data Management
  - NC Data Transfer
  - Tool Data Communication

Manufacturer/Service Documentation

- *SINUMERIK SIMODRIVE* — Description of Functions SINUMERIK **Safety Integrated**
- *SINUMERIK* 840D — Description of Functions **Digitizing**
- *SINUMERIK SIMODRIVE* — **Installation & Start-Up Guide *)**
  - 810D
  - 840D/611D
  - HMI
- *SINUMERIK SIMODRIVE* 840D/840Di 810D 611D — **– Lists *)**
- *SINUMERIK SIMODRIVE* 840D 611D — Description of Functions **Linear Motor**
- *SINUMERIK SIMODRIVE* 840D 611D — Description of Functions **– Hydraulics Module** **– Analog Module**
- *MOTION CONTROL SYSTEMS* — **EMC** Guidelines

Electronic Documentation

- *SINUMERIK SIMODRIVE* 840D/840Di/810D 611, Motors — **DOC ON CD *)** The SINUMERIK System

Manufacturer/Service Documentation

- *SINUMERIK* 840D/840Di/810D — Description of Functions **ISO Dialects** for SINUMERIK
- *SINUMERIK* 840Di — **Manual** (HW + Installation and Start-Up)
- *SINUMERIK* 840D/810D — Description of Functions **Remote Diagnosis**
- *SINUMERIK* 840D/840Di/810D — **Manual @ Event**

---

*) These documents are a minimum requirement

Siemens AG
Automation and Drives
Motion Control Systems
P.O. Box 3180, D-91050 Erlangen
Germany

www.siemens.com/motioncontrol

Order No.: 6FC5298-6AC00-0BP3

Printed in Germany