SIEMENS

SINUMERIK 840D sl

HMI-Embedded

Operating Manual

Valid for

PLC	*Software version*
SINUMERIK 840D sl/840DE sl	
Software	*Version*
NCU System Software for 840D sl/840DE sl	1.5
with HMI Embedded	7.5

01/2008

SINUMERIK® Documentation

Printing history

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

The status of each version is indicated by the code in the "Remarks" columns.

Status code in the "Remarks" column:

A .... New documentation.
B .... Unrevised reprint with new order number.
C .... Revised edition with new status.

Issue	Order No.	Comment
02.01	6FC5298-6AC00-0BP0	A
11.01	6FC5298-6AC00-0BP1	C
11.02	6FC5298-6AC00-0BP2	C
03.04	6FC5298-6AC00-0BP3	C
08/2005	6FC5398-1AP10-0BA0	C
11/2006	6FC5398-1AP10-1BA0	C
01/2008	6FC5398-1AP10-2BA0	C

Registered trademarks

All designations with the trademark symbol ® are registered trademarks of Siemens AG. Other designations in this documentation may be trademarks whose use by third parties for their own purposes may infringe the rights of the owner.

Exclusion of Liability

We have checked that the contents of this document correspond to the hardware and software described. Since deviations cannot be precluded entirely, we cannot guarantee full agreement. The information contained in this document is, however, reviewed regularly and any necessary changes will be included in the next edition.

Copyright © Siemens AG 1995 - 2008
Order No. 6FC5398-1AP10-2BA0

Siemens AG 2008.
We reserve the right to make technical changes

Preface

Structure of the Documentation

The SINUMERIK documentation is organized in 3 parts:

- General documentation
- User documentation
- Manufacturer/Service Documentation

An overview of publications, which is updated monthly and also provides information about the language versions available, can be found on the Internet at:
http://www.siemens.com/motioncontrol
Follow the menu items "Support" → "Technical Documentation" → "Overview of Publications".

The Internet version of DOConCD (DOConWEB) is available at:
http://www.automation.siemens.com/doconweb

Information about training courses and FAQs (Frequently Asked Questions) can be found in internet under::
http://www.siemens.com/motioncontrol under menu item "Support"

Validity

These Operating Instructions are valid for HMI Embedded SW 7.5

Technical Support

If you have any questions, please contact our hotline:

Europe and Africa:
A&D Technical Support
Tel.: +49 (0) 180 / 5050 - 222
Fax: +49 (0) 180 / 5050 - 223
Internet:http://www.siemens.com/automation/support-request

America:
A&D Technical Support
Tel.: +1 423 262 2522
Fax: +1 423 262 2289
E-mail: mailto:techsupport.sea@siemens.com

Asia/Pacific:
A&D Technical Support
Tel.: +86 1064 719 990
Fax: +86 1064 747 474
E-mail: mailto:adsupport.asia@siemens.com

© Siemens AG 2008 All rights reserved.

Note

National telephone numbers for technical support are provided under the following Internet address:

http://www.siemens.com/automation/service&support

SINUMERIK Internet address

http://www.siemens.com/sinumerik

Standard scope

This Operator's Guide describes only the functionality of the standard version. Additions or revisions made by the machine manufacturer are documented by the machine manufacturer.

Other functions not described in this documentation might be executable in the control. However, no claim can be made regarding the availability of these functions when the equipment is first supplied or in the event of servicing.

Qualified persons

The associated device/system may only be set-up and operated in conjunction with this documentation. The equipment / system may only be commissioned and operated by **qualified personnel**. For the purpose of the safety information in this documentation, a "qualified person" is someone who is authorized to energize, ground, and tag equipment, systems, and circuits in accordance with established safety procedures.

Designated use

WARNING

Note the following:
The device may be used only for the applications described in the catalog and in the technical description, and only in combination with the equipment, components and devices of other manufacturers where recommended or permitted by Siemens. To ensure trouble-free and safe operation of the product, it must be properly transported, stored, and installed, and maintained and operated with care.

Structure of the descriptions

All functions and operating options have been described according to the same internal structure as far as this is meaningful and practicable. The various levels of information have been organized such that you can selectively access the information you need for the task in hand.

© Siemens AG 2008 All rights reserved.

Explanation of symbols

Function

The theoretical section is primarily intended as learning material for the NC entry-level user and includes important information to assist the user to understand the operator functions.
You should work through the manual at least once to get an idea of the operational scope and capability of your SINUMERIK control.

Sequence of operations

This section contains the sequence of keys required for operation at a glance. If inputs have to be made at individual stages of the sequence or if you require additional information, you will find this next to the key illustrations.

Notes

This symbol appears in this documentation whenever it is necessary to draw your attention to an important item of information.

Machine manufacturer

For safety reasons, some functions are disabled to protect them from unauthorized access. The machine manufacturer can customize or modify the described functionality. Please comply fully with the instructions of the machine-tool manufacturer.

Additional information

For safety reasons, some functions are disabled to protect them from unauthorized access. The machine manufacturer can customize or modify the described functionality. Please comply fully with the instructions of the machine-tool manufacturer.

References

This symbol appears whenever specific information can be found in other documentation.

© Siemens AG 2008 All rights reserved.

Ordering data option

In this documentation, you will find this symbol with a reference to an ordering data option. The described function can only run if the control contains the designated option.

Safety information

This Manual contains information which you should carefully observe to ensure your own personal safety and the prevention of material damage. The notices referring to your personal safety are highlighted in the manual by a safety alert symbol, notices referring to property damage only, have no safety alert symbol. Depending on the hazard level, warnings are indicated in a descending order as follows:

Danger

Indicates that death or severe personal injury **will** result if proper precautions are not taken.

WARNING

indicates that death or severe personal injury **may** result if proper precautions are not taken.

Caution

means that there can be slight physical injury if the corresponding safety measures are not followed.

Caution

without a safety alert symbol, indicates that property damage can result if proper precautions are not taken.

Notice

indicates that an unintended result or situation can occur if the corresponding information is not taken into account.

If multiple levels of hazards can occur, the warning is always displayed with the highest possible level. If a warning notice with a safety alert symbol is to indicate physical injury, the same warning may also contain information about damage to property.

■

© Siemens AG 2008 All rights reserved.

Contents

© Siemens AG 2008 All rights reserved.

© Siemens AG 2008 All rights reserved.

© Siemens AG 2008 All rights reserved.

© Siemens AG 2008 All rights reserved.

© Siemens AG 2008 All rights reserved.

© Siemens AG 2008 All rights reserved.

© Siemens AG 2008 All rights reserved.

© Siemens AG 2008 All rights reserved.

Introduction

© Siemens AG 2008 All rights reserved.

1.1 The SINUMERIK 840D sl

General

SINUMERIK 840D sl is a CNC control system (**C**omputerized **N**umerical **C**ontrol) for machine tools.
The SINUMERIK 840DE sl is available as an export version for use in countries where approval is required.

The heart of the SINUMERIK 840D sl is the Numerical Control Unit - NCU 710. It combines NCK, PLC, HMI, control and communications functions.

You can use the front control panel to implement the following basic functions for a machine tool:

- Creation and adaptation of part programs
- Execution of part programs
- Manual control
- Reading in and reading out of part programs and data
- Editing of data for programs
- Displaying and troubleshooting alarms
- Editing of machine data
- Establishing communications connections between one or more TCUs.

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

- Display units such as screen, LEDs, etc.
- Operator controls such as keys, switches, hand wheels, etc.

These operating instructions describe the operation of the
HMI Embedded software, SW 7.5

The HMI Embedded software is on the CompactFlash Card on delivery of the NCU.

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

© Siemens AG 2008 All rights reserved.

1.2 Switching the control on/off

Function

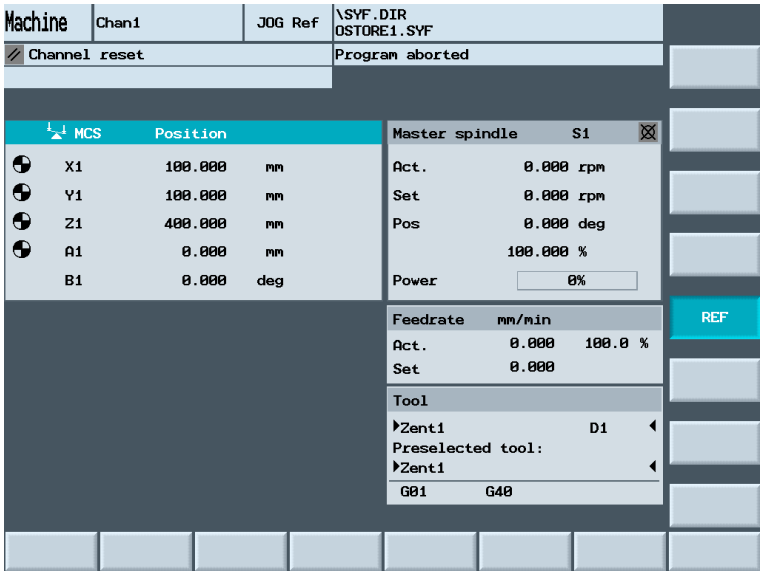
Switching the control ON

A variety of methods can be employed to switch on the power supply to the control system or to the whole plant.

Machine manufacturer

Please follow the machine manufacturer's instructions!

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 below for switching off the control or the entire system!

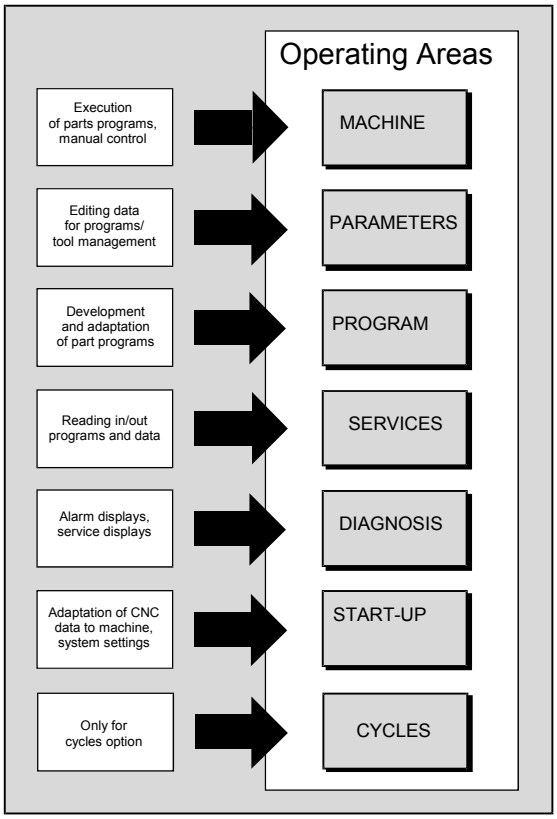
Machine manufacturer

Please follow the machine manufacturer's instructions!

© Siemens AG 2008 All rights reserved.

1.3 Operating areas

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

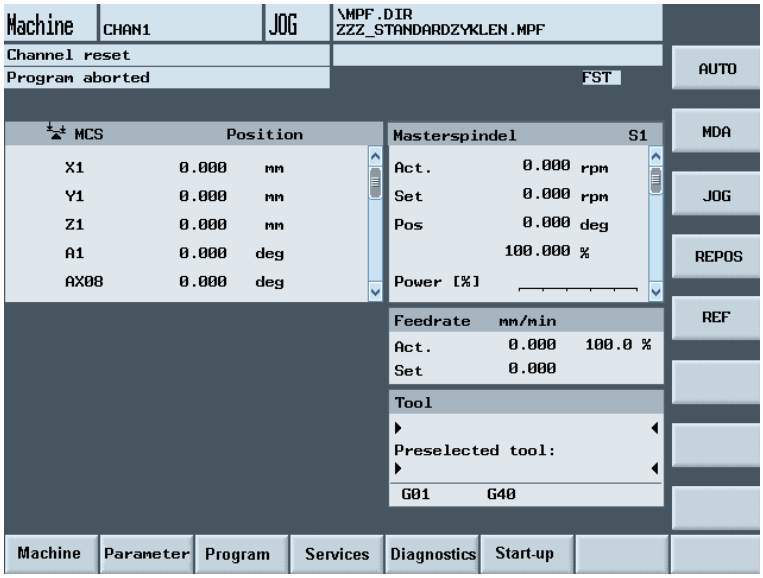


Sequence of operations

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.

© Siemens AG 2008 All rights reserved.



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.

■

© Siemens AG 2008 All rights reserved.

© Siemens AG 2008 All rights reserved.

Operator Components/Operating Sequences

© Siemens AG 2008 All rights reserved.

2.1 Operator panels

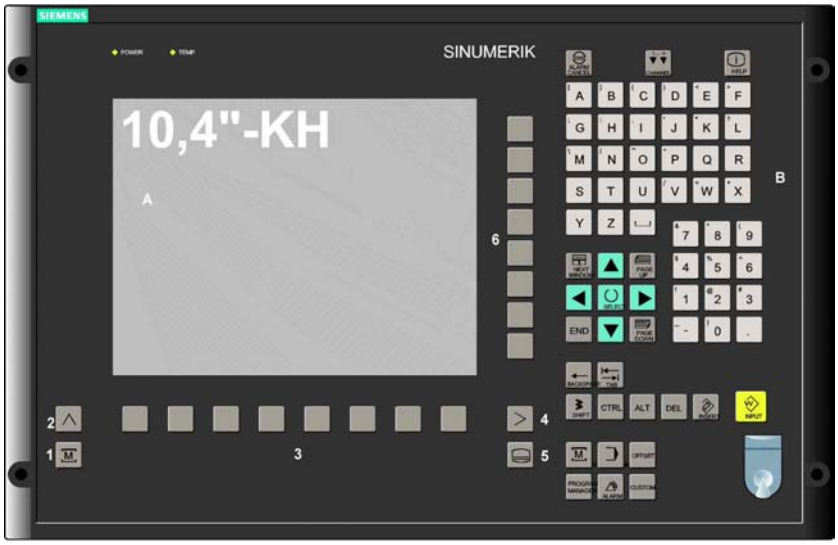
Example

The controls available for operation of the SINUMERIK controller and the machine tool are described below using the SINUMERIK OP 010 operator panel as an example.

Operating panel OP 010

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)



The keys are described in the following sections.

© Siemens AG 2008 All rights reserved.

2.1.1 Keys on the control panels

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 higher-level menu. Recall closes a window.

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

MENU SELECT
You can call the main 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 main menu branches into the following operating areas:

1. Machine
2. Parameter
3. Program
4. Services
5. Diagnostics
6. Commissioning

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 through to channel n).
When a "Channel menu" is configured, all existing communication links plus the associated channels are displayed on softkeys.
(See also Section "Switch over channel")

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

© Siemens AG 2008 All rights reserved.

SHIFT
Depress the Shift key to enter the upper 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 beginning.
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, space character

BACKSPACE
- Delete the value in the input field from the right.
- In insertion mode, it deletes the character after the cursor.

DEL
- Delete the value in the parameter field
- In insertion mode, it deletes the character marked by the cursor.

INSERT
- Switch over to edit mode in tables and input fields (in this case, the input field is in insert mode) or
- UNDO function on table elements and input fields (the value is not validated when you exit a field using the edit key; instead it is reset to the previous value = UNDO).
- Activate pocket calculator mode

TAB
Tab key

© Siemens AG 2008 All rights reserved.

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

Navigate 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

- Navigate 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 input fields and selection lists labeled with this key symbol.
- Activate/deactivate a field:

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

- Switch Select mode on and off in the editor

© Siemens AG 2008 All rights reserved.

Cursor to the right

- Navigate between different fields or lines.
- Opens a directory or program.

END

- With this key, the cursor is moved to the end of the line in the page opened in the editor.
- Rapid positioning of the cursor on a group of related input fields.

Cursor down

Navigate 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 window that is active. The scroll bar indicates which part of the program/document/... is selected.

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

OFFSET = (only OP 010 and OP 010C)
Tool management
Opens the "Tools/Offsets" 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 (OP 010 and OP 010C only)
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.

© Siemens AG 2008 All rights reserved.

2.2 Machine control panels

Standard turning machines/milling machines

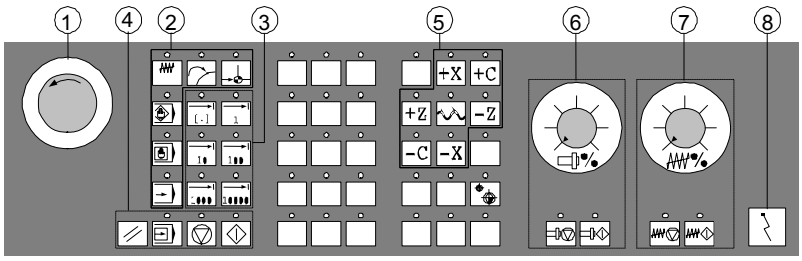
Actions on the machine tool, for example traversing the axes or program start, can only be initiated via a machine control panel.

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.

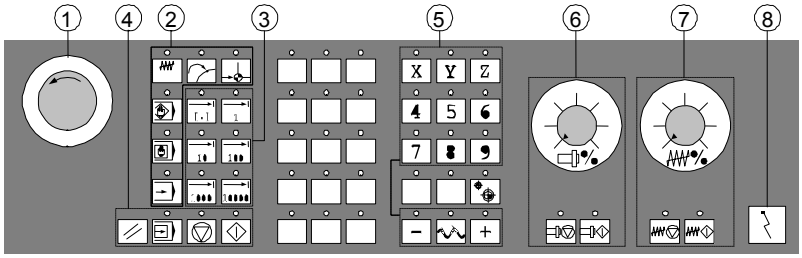
The machine control panel described in this document is the panel supplied by SIEMENS. If you are using another machine control panel, please consult the operating instructions of the machine-tool manufacturer.

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

1 EMERGENCY STOP button
2 Operating modes (with machine functions)
3 Incremental mode
4 Program control instructions
5 Direction key with rapid traverse override
6 Spindle control
7 Feedrate control
8 Key-operated switch

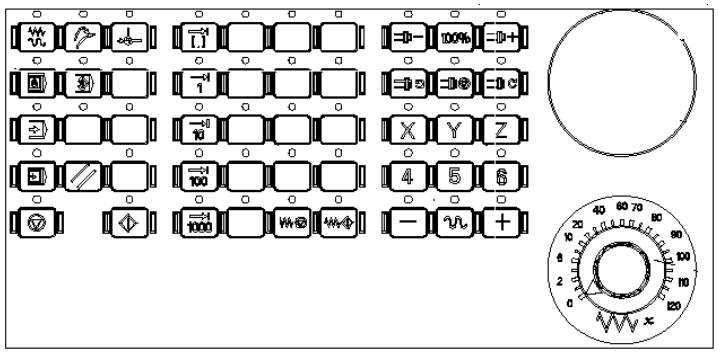


Machine control panel for turning machines



Machine control panel for milling machines

Machine control panel OP 032S

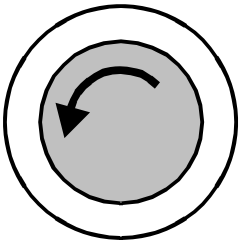


Machine control panel OP 032S

© Siemens AG 2008 All rights reserved.

2.3 Keys on the machine control panel

2.3.1 Emergency Stop key



Press this red key in emergency situations:
1. if life is at risk
2. When there is a danger of the 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 follow the instructions of 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. Axis travel in jog mode 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 Automatic” operating mode.
Control of machine through execution of a block or a sequence of blocks. The blocks are entered on the operator panel front.

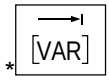
Automatic
Select “Machine Automatic” operating mode.
Control of machine through automatic execution of programs.

© Siemens AG 2008 All rights reserved.

Inc keys

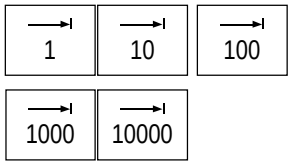
You can activate the Inc functions in conjunction with the following modes:

- "JOG" operating mode
- "MDA/Teach-in" operating mode



*

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, 1,000, 10,000 increments.

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

Please refer to the machine manufacturer's instructions.

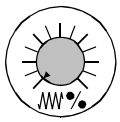
Machine functions

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

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

© Siemens AG 2008 All rights reserved.

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

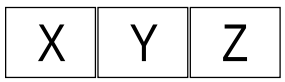
- Execution of the current program is stopped
- The axis drives are brought to a standstill under control,
- The associated LED illuminates as soon as feed stop has been accepted by the control,
- FST (= feed stop) appears in the header (program control display)

Example:
– In "MDA" mode, an error is detected during execution of a block.
– A tool change is to be carried out.

Feed Start

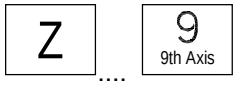
If you press the "Feed start" key:
- The part program is continued in the current block,
- The feedrate is accelerated to the value specified by the program,
- The associated LED lights up as soon as feed start has been accepted by the control.

Axis keys (for turning machines):



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

Axis keys (for milling machines):



Select the axis (X ... 9) for traversing

in the positive direction by pressing the "+" key or

in the negative direction by pressing the "-" key.

© Siemens AG 2008 All rights reserved.

Rapid
Traverses axis at rapid traverse (fastest speed).

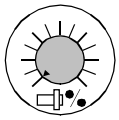
Machine manufacturer

- 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 follow the machine manufacturer's instructions!

WCS/MCS
Switches 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 on main screen).

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 lights up as soon as "Spindle stop" is reached.

Example:

- To change a tool
- To enter S, T, H, M functions during setup

© Siemens AG 2008 All rights reserved.

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 edited by the machine-tool manufacturer to suit the specific 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-operated switches

SIEMENS key switch

The keylock 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-operated 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-operated switch has three different colored keys which can be removed in the specified positions:

© Siemens AG 2008 All rights reserved.

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

Lowest
Access authorization

Highest
Access authorization

Changing access authorization

The screen is not automatically updated after a change in access authorization (e.g., when the key-operated 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-operated switch positions are not evaluated during commissioning.

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-operated switch positions are irrelevant.

References

Commissioning Manual, Commissioning Base software and HMI-Embedded; Commissioning HMI-Embedded (IM2): Protection levels

© Siemens AG 2008 All rights reserved.

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,

- Stop is displayed on the screen in the cycle (in the program control line)
- The message "Stop: Block completed in single block" is displayed in the channel operational message line (in interrupt state).
- the current block of the part program is only processed once you press the "Cycle Start" key,
- Program processing is stopped after one block is executed.
- 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.

© Siemens AG 2008 All rights reserved.

Reset (Reset)

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

References Function Manual, Basic Functions; Axes, Coordinate System, Frames (K1): Mode group, channel, program operation mode

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	1...n 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

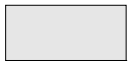
Caution

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.

© Siemens AG 2008 All rights reserved.

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.

1. Operating areas
2. Channel status
3. Program status
4. Channel name
5. Alarm and message line
6. Mode
7. Program name/path of the selected program
8. Channel operational messages
9. Program control
10. Additional information (help)
    - i You can display information by pressing 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.
    The unit for positional data is preceded by the diameter symbol ∅ in work panes if the axis is currently the transverse axis and the tool coordinate system is set. If diameter programming is disabled with DIAMOF, the symbol preceding the unit is no longer visible.

© Siemens AG 2008 All rights reserved.

12. User response line with user information
    Here user information is displayed for the selected function (if available).

13. Focus
    The selected pane is highlighted by a 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).

© Siemens AG 2008 All rights reserved.

2.4.2 Global machine status display

1 Operating areas	The currently selected operating area is displayed (Machine, Parameters, Program, Services, Diagnostics, Commissioning).
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 • Instructions that were programmed in the part program using 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: No read-in enable 9 Wait: No feed enable 10 Wait: Remaining dwell time: ... secs

© Siemens AG 2008 All rights reserved.

11 Wait: No aux. funct. ackn.
12 Wait: No axis enable
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

- Activation of position control
- Requesting of zero marker
- Reference point approach active
- Parameter block change active
- Change of measuring system active
- Measurement on-the-fly active
- Closed-loop controller enable removed
- Axis/spindle disable, status changes

© Siemens AG 2008 All rights reserved.

39 Wait: AXCT axis/spindle 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 synchronization of MD data (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
55 channel
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
61 master-value coupling
Waiting for axis replacement: Axis is currently coupled-motion
62 axis
Waiting for axis replacement: Axis is currently coupled slave axis

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

© Siemens AG 2008 All rights reserved.

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 Function Manual, Basic Functions; Axes, Coordinate System, Frames (K1): Program control

DRY
DRY run feedrate

If you select this softkey, all traversing motions will be performed with the feedrate value specified via the "Dry run feed" setting data. The dry run feedrate function replaces the programmed travel commands.

ROV
Rapid traverse override

The feedrate override switch will also act on the rapid traverse override.

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 activated, special function M01 (from the part program) is **not** taken into account.

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

If "Program test" is selected, the output of setpoints to axes and spindles is disabled. The setpoint 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.

© Siemens AG 2008 All rights reserved.

2.5 General operating sequences

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

Sequence of operations

"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.

Program overview

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:

Program selection

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

© Siemens AG 2008 All rights reserved.

2.5.2 Selecting and opening a 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 information supplied by 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.

Sequence of operations

2nd file

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.

© Siemens AG 2008 All rights reserved.

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:

- Newly created program enabled automatically after it is closed.
- Hide the "Line feed“ block end,
  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 time for positioning in a large file (e.g. in mold construction). Enter the size of the file in KB from which you wish to disable line checking.

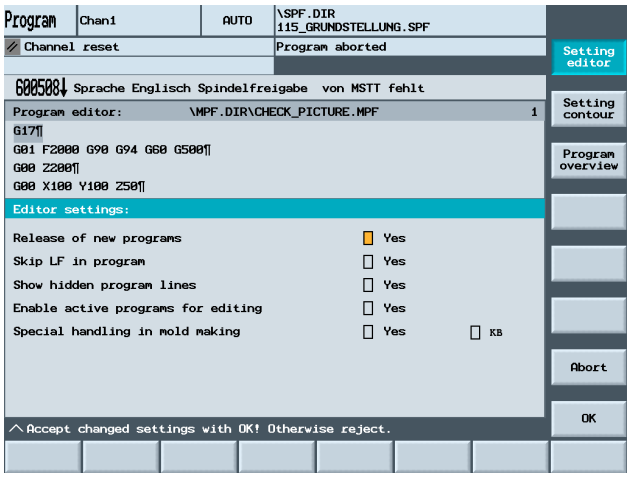
Sequence of operations

Settings

Press the horizontal softkey "Settings"

Editor Settings

Press the vertical softkey “Editor Settings” to open the window for the Editor settings:



© Siemens AG 2008 All rights reserved.

2.5.4 Contour programming

Contour Settings

Press the “Contour settings” softkey to open the window and make the settings for contour programming.

- Enter in the line below "Text output at the end of the contour programming" the text that is to appear after each contour, e.g. "Contour end"
- Setting the technology: Select “Turning”
- Display the softkeys for contour elements as symbols or text.

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

2.5.5 Summary of ranges

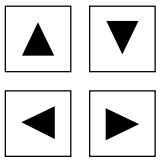
Program overview

Press the “Program overview” softkey to select one of the available drives, which become the default selection for operations such as saving.

2.5.6 Toggling between menu windows

You can toggle between individual menu windows with the “NEXT WINDOW” key. 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 a window contains more information than can be shown at once, a scroll bar is shown and you can scroll through the contents of the window with “PAGE UP” and “PAGE DOWN”.



Position cursor in menu window:
You can position the cursor at the desired point in the menu window with the direction keys.

© Siemens AG 2008 All rights reserved.

2.5.7 Edit 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 option 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-option button/option button (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.

© Siemens AG 2008 All rights reserved.

2.5.8 Confirming/canceling input

Confirming input:

OK

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

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 "Edit" 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.

2.5.9 Editing a part program in the ASCII editor

Function

The editor provides you with the following functions:

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

© Siemens AG 2008 All rights reserved.

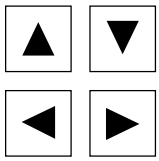
Additional information

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

Sequence of operations

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

You have selected the file that you want to edit in the directory and you press 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 BACKSPACE key to delete the character to the left of the cursor.

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

© Siemens AG 2008 All rights reserved.

Vertical softkeys

Overwrite

Overwrite

The cursor toggles between insert and overwrite modes.

Select Block

Select 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.

Select block or SELECT

You can cancel select mode with softkey "Select 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

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

© Siemens AG 2008 All rights reserved.

The following search methods are available:

Start of Program or CTRL NEXT WINDOW

- Search 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

Find

Go to

- Search for a particular character string with "Find"
- Or proceed to a particular NC block with "Go to.."

Enter the block number you are looking for.

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

OK

Press the "OK" softkey 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 press “Abort” the positioning 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 the "Find Next" softkey 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 press “Abort”, the Find + Replace process is stopped and the window closed. You are in "Edit mode" again.

© Siemens AG 2008 All rights reserved.

Save file

Modifications are saved in the file in the editor by pressing the "Save file" softkey.

Additional information

Please note that the changes to programs stored in the NCK 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 programs with the appropriate parameter settings.

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 parameter settings via the Drilling, Milling, Turning vertical softkeys.

Enter the new values for the cycle parameters.

OK

The part program block is inserted in the part programs with the appropriate parameter settings.

Example:

CYCLE81 (110, 100, 2, 35)

References

Programming Manual Cycles

Simulation

Simulation

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

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.

© Siemens AG 2008 All rights reserved.

Recompil.

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 parameter settings.

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 follow the machine manufacturer's instructions!

References

Commissioning Manual, Commissioning Base software and HMI-Embedded; Commissioning HMI-Embedded (IM2): Coordinate systems

2.5.10 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.
When a "Channel menu" is configured, all existing communication links are displayed on softkeys.

Channel statuses

The following three channel statuses 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.

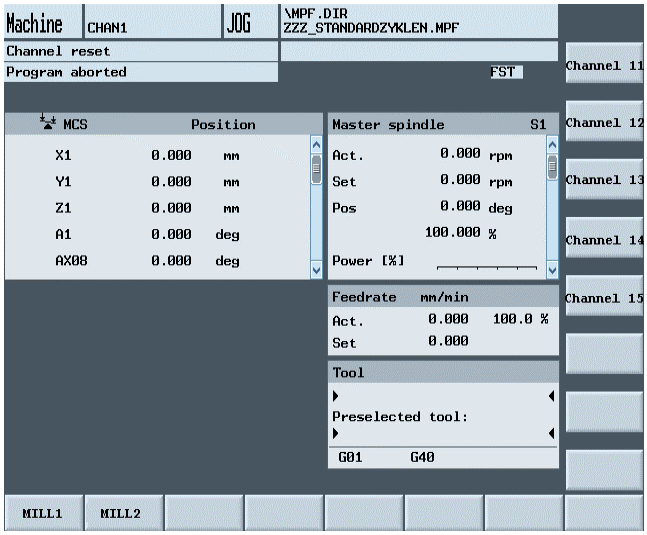
© Siemens AG 2008 All rights reserved.

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 different levels:
1. Switch to next channel.
2. Switch over configured channel group/channels (1 NCU).

Function
You can set up a link in any operating area between the HMI unit and the connected NCK/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.

© Siemens AG 2008 All rights reserved.

Additional information

- A channel selection is triggered by selection of a vertical softkey.
- Channels that are configured in the channel menu, but defined as a channel gap in the corresponding NCK, will not be displayed.

2.5.11 Pocket calculator

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

You can switch to Pocket calculator mode with the equals key.

In this state, if you enter a basic arithmetic symbol (+, -, /, *) followed by a value (e.g. 13.5) and

then press the input key, the next value entered is calculated with the preceding 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.

© Siemens AG 2008 All rights reserved.

2.5.12 Entering fittings

Function

The letter "F" ("f") is used to identify a fitting. 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 = fitting
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!)

Limitations:

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,
7. 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 / lower case:

Because the control panel has lower-case lettering and there is not option for shifting to upper case, an external keyboard is required for input of upper and lower case.

When you have finished, press the ENTER key to confirm your input.

© Siemens AG 2008 All rights reserved.

2.5.13 “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

Commissioning Manual, Commissioning Base software and HMI-Embedded; Commissioning HMI-Embedded (IM2): "Blue screen" error status screen

■

© Siemens AG 2008 All rights reserved.

Example of Operation

© Siemens AG 2008 All rights reserved.

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	1.2
	• Reference point approach	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 data interface	6.4 7.5
	• Select a part program	4.6
	• Execute a trial program run (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 4.6 8.1
	• Optimize a part program	4.2
Machining the workpiece	• Use tool and execute machining program	5.2
Store a program	• Save a part program - on external storage devices	6.7 4.6, 7.5

■

© Siemens AG 2008 All rights reserved.

Machine operating area

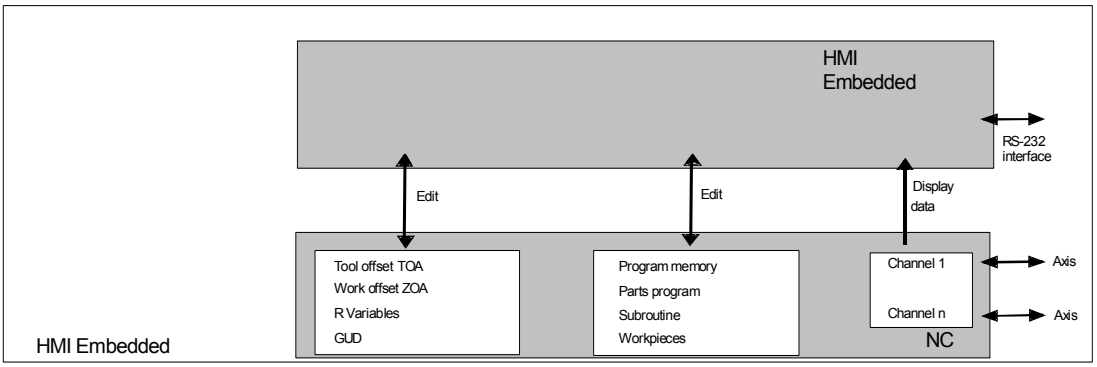
© Siemens AG 2008 All rights reserved.

4.1 Data structure of the NCK control

Function

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

HMI-Embedded The data are always in the NCK and are also directly modified there.



4.1.1 Operating modes and machine functions

Function

The Machine operating area includes all functions and influencing variables 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).

© Siemens AG 2008 All rights reserved.

- 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.
- Teach In:
  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., part programs are selected, started, corrected, selectively controlled (e.g., single block) and executed in Automatic.

Select machine area

Machine

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 "Machine" operating area and in "Jog" mode. (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 (repositioning according to 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.

© Siemens AG 2008 All rights reserved.

Preparation for production

To start actual production, some preparatory measures must be taken:

1. Set up the tools and workpiece
2. Traverse the tools/workpiece to the start position specified in the setup plan
3. Download the part program to the control memory
4. Check/enter the work offsets
5. Check/enter the tool offsets

4.1.2 Operating mode group and channels

Function

Every channel behaves like an independent NCK 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 "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 (machining)
	Channel 4 (transport)

Technologically-related channels can be combined to form a mode group.
Axes and spindles of the same mode group can be controlled by one or more channels.
An operating mode group is in one of "Automatic", "Jog" or "MDA" operating modes, i.e., several channels of an operating mode group can never assume different operating modes.

© Siemens AG 2008 All rights reserved.

4.1.3 Selecting / 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

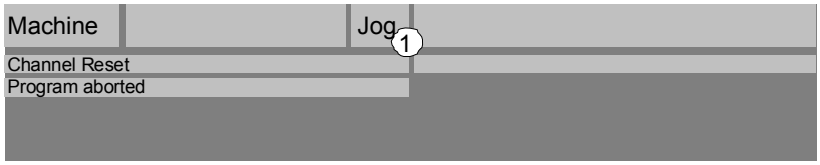
Whether the requested mode can be accessed and the manner in which it is accessed can be configured for a specific machine in the PLC program.

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.

Sequence of operations

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



1 = operating mode

Press one of the keys on the machine control panel to select the modes:

- Jog

- MDA

- Automatic

or
Press the “MENU SELECT” range switching key and the corresponding vertical softkey:

© Siemens AG 2008 All rights reserved.

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.

Additional information

The main screen 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 for trained personnel only on safety grounds. To provide this type of protection, the control system offers a facility for disabling or enabling mode changes.

References

Function Manual, Fundamentals; Mode group, Channel, Program operation mode (K1)

© Siemens AG 2008 All rights reserved.

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.

Sequence of operations

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

Prerequisite:

- No alarms are pending.
- The program is selected.
- Feed enable is active.
- Spindle enable is set.

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

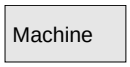
Stop/abort part program:

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

The current program is aborted.

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

Repositioning:

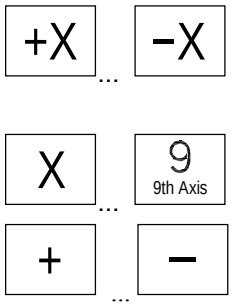


Select the "Machine" operating area.

Select "Jog" mode.

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

© Siemens AG 2008 All rights reserved.



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 Displaying a 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.

Sequence of operations

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

Program level

When you press the "Program level" softkey, the window headed "Program level" 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 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.

© Siemens AG 2008 All rights reserved.

4.2.3 Toggle between 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 work 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., work 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 (= Work) 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	
C1	

© Siemens AG 2008 All rights reserved.

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 follow the machine manufacturer's instructions!

Sequence of operations

"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 press the "Act. val. WCS" softkey, 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 NCK program refer to the WCS.

Zoom act. val.

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

- 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 measurement 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).

© Siemens AG 2008 All rights reserved.

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 Function Manual, Fundamentals; Axes, Coordinate Systems, Frames (K2)

4.2.4 Actual value display: Settable zero point system, SZS

Function

You can set which data must be displayed in the actual value display via MD:

- 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)

For configuring instructions, see:

References Commissioning Manual, Commissioning Base software and HMI-Embedded; Commissioning HMI-Embedded (IM2): Work offset

4.2.5 Displaying 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.

Sequence of operations

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

© Siemens AG 2008 All rights reserved.

Axis feedrate

Press the "Axis feedrate" softkey:

- With "Machine" the feed window is displayed with the current feedrates and distance-to-go information as well as the associated override.
- With "Work" the feed window for the axes taking part in the interpolation is displayed with the current feed and the distance-to-go information with path override; for the remaining axes the current feed and distance-to-go information is displayed with single-axis override.

PAGE DOWN or PAGE UP

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

4.2.6 Displaying spindles

Function

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

Sequence of operations

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

Spindle

The “Spindle” window section is activated and shows the following data:

- Spindle speed setpoint and actual value
- Spindle position
- Spindle override switch position
- Spindle power (%)
- Spindle load (for max. 2 analog spindles)

PAGE DOWN or PAGE UP

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

Additional information

- The "Spindle" window is only displayed 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.

© Siemens AG 2008 All rights reserved.

4.2.7 Displaying auxiliary functions

Function

Auxiliary functions that are active in the selected channel are displayed.
Modal M functions remain active until they are deleted or overwritten by another command.
M functions are not merely displayed block-by-block, but also remain visible for as long as they are active.

Sequence of operations

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

Auxiliary function

The “Auxiliary function” window section is shown.

The statuses of M functions are displayed as follows:

M08 M function active

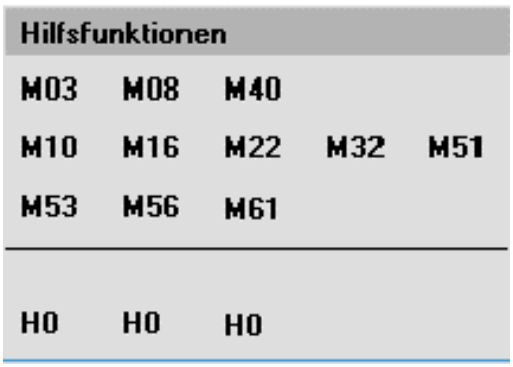
M40 M function retrieved during search, but not yet output to PLC

M22 M function output to PLC but not yet acknowledged (yellow text on black background signifies: Wait)

Window division

4 lines are displayed:
1. Row: Standard M functions:
   M03/M04/M05/M19, M7/M8, M40/M41/M42/M43/M44/M45
2. Row: customized, grouped together M functions
3. Row: non-grouped M functions
4. Row: H functions

The lines are oriented toward the actual value positions.
A maximum of 5 M functions and 3 H functions can be displayed on each line.
Example:



References Function Manual Fundamentals

© Siemens AG 2008 All rights reserved.

4.2.8 Display transformations, G functions, and swivel data record

Function

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

Sequence of operations

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

Transform./ G functions

The window "Transformation/ 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.

Additional 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 Function Manual Fundamentals

Swivel data status display

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

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

References Programming Manual Cycles: Swiveling

© Siemens AG 2008 All rights reserved.

4.2.9 Handwheel

Function

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

Sequence of operations

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

Handwheel

The “Handwheel” window section is activated.

▲ ◀ ▶ ▼

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

SELECT

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).

Handwhl	Axis	Enable
1	X1	X
2	Y1	
3	Z1	

SELECT

Every time you press the "SELECT" key in the "Enable" 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).

OK

Confirm these settings with the "OK" softkey.

Machine manufacturer

The machine manufacturer is responsible for the design of handwheels. Operation may therefore differ from the explanation above.
Please follow the machine manufacturer's instructions!

© Siemens AG 2008 All rights reserved.

4.2.10 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! The protection zones and software limit switches are only reactivated following another reference point approach.

Additional information

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

Machine manufacturer Please follow the machine manufacturer's instructions!

Sequence of operations

"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 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.

OK

Confirm the settings with the “OK” softkey.

Additional information

Machine manufacturer The "Preset" function can be disabled by means of protection levels (keylock switch position).

© Siemens AG 2008 All rights reserved.

4.2.11 Setting the 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 as “Preset”.

Machine manufacturer

Please follow the machine manufacturer's instructions!

References

Commissioning Manual, Commissioning Base software and HMI-Embedded; Commissioning HMI-Embedded (IM2)

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

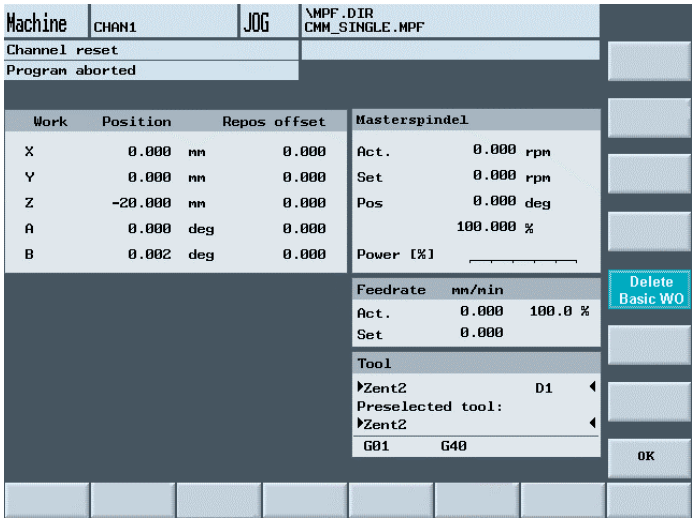
Sequence of operations

Set actual value

1 ... 9

INPUT

The new position setpoint 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".

© Siemens AG 2008 All rights reserved.

4.2.12 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 part program start or mode change are disabled for the duration of the switchover.

Sequence of operations

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

Press the “ETC” key followed by

the “Switch mm > inch” softkey.

Switch mm > inch

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

The switchover is displayed with a corresponding message on the user interface, e.g.:
“Notice! Measuring system is switched from metric to inches”

References

Function Manual, Fundamentals; Velocities, Setpoint-Actual Value Systems, Closed-Loop Control (G2) Metric/inch measuring system
Commissioning Manual, Commissioning Base software and HMI-Embedded; Commissioning HMI-Embedded (IM2)

© Siemens AG 2008 All rights reserved.

4.3 Approaching a reference point

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/handwheel) 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.

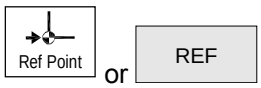
Caution

- 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 follow the machine manufacturer's instructions!

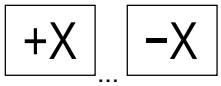
Sequence of operations

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

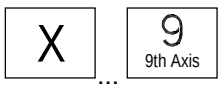


The "Ref Point" machine function is selected.

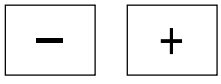
Turning machine:
Press the "Axis" keys.



Milling machine:
Select the axis to be traversed and



then press the "+" or "-" key.



© Siemens AG 2008 All rights reserved.

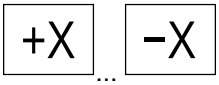
The selected axis moves to the reference point. The direction and sequence is defined by the machine manufacturer in the PLC program.
If you have pressed the wrong direction key, the action is not accepted and the axes do not move.
The display shows the reference point value.

No symbol is displayed for axes that are not referred to a reference point.

○ This symbol appears for axes that must be referenced.

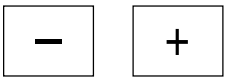
◒ This symbol is displayed next to the axis when the reference point has been reached.

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



then 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.

© Siemens AG 2008 All rights reserved.

Machine manufacturer

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

Additional 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 mode

4.4.1 Function and main screen

Function

In manual mode you can:

1. Synchronize the control measuring system with the machine (reference point approach),
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 being interrupted.

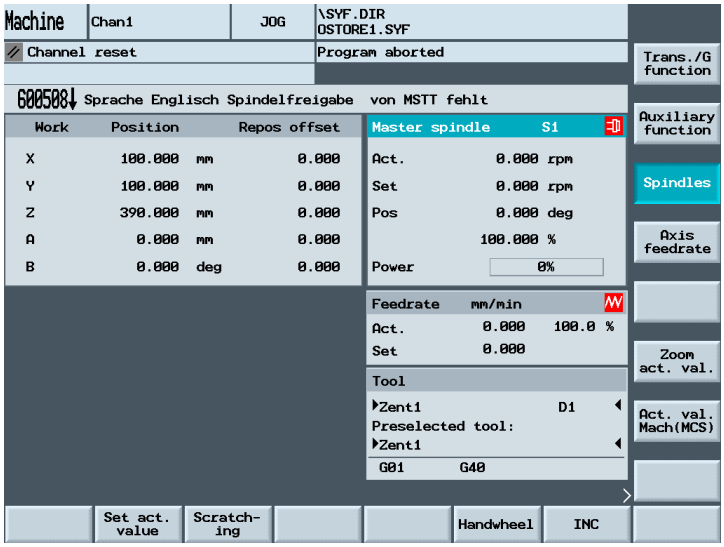
or JOG

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

followed by the "Jog" key.

The "Jog" main screen contains values relating to position, feedrate, spindle and tool.

© Siemens AG 2008 All rights reserved.



Explanation of the Jog main screen

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 geometry 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 follow the machine manufacturer's instructions!

- If the axis identifier can only be displayed in an abbreviated form, this is indicated 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.

Location
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.

© Siemens AG 2008 All rights reserved.

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
U/min

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:

- The "Path section" and "Stroke rate" are displayed in reverse video if you have programmed the automatic block division with "Path section length" or "Number of path sections".
- The "delay time" can only be displayed if you have set "Punching with delay time".

Feedrate
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.

© Siemens AG 2008 All rights reserved.

Tool

Tool window
Display of active tool offset (e.g., D1), tool currently in use (T no.), preselected tool (on milling machines) plus currently active motion 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

Traverse rate

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

Sequence of operations

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

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

[VAR]

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

1 ... 10000

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

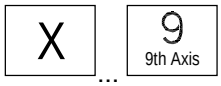
Turning machine:
+X ... −X
Press the "Axis" keys.

If necessary, set the rate with the override selector.

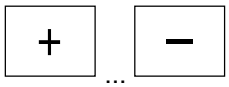
Rapid

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

© Siemens AG 2008 All rights reserved.



Milling machine:
Select the axis to be traversed and



then 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).

Additional information

- When the control is switched on, the axes can be traversed right up to the limits of the machine as the reference points have not yet been approached and the axes 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: dimension

Function

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

Sequence of operations

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

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

Enter the desired “INC-Var” increment size

© Siemens AG 2008 All rights reserved.

Abort

Jump 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 selected axis in the appropriate direction in accordance with the increments set above (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).

Sequence of operations

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

Repos or REPOS

Select machine function "Repos".

+X ... −X

Turning machine:
Press the "Axis" keys.

X ... 9 9th Axis

Milling machine:
Select the axis to be traversed and

− ... +

press the "+" or "–" key.

© Siemens AG 2008 All rights reserved.

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

WARNING
The rapid traverse override key 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 NCK, you must enable or disable the function depending on the keyswitch position in the "Approach reference point" operating mode.

User agreement can only be granted if keylock switch position 3 or higher applies in respect of the access rights.

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

Sequence of operations

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

User agreement

Press the "User agreement" softkey.
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 user agreement.

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 the axis is referenced. If not, error message "Please reference axis first" is displayed. User agreement cannot be activated for the axis until it has been referenced.

© Siemens AG 2008 All rights reserved.

Additional information

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

References

For more information please refer to:
Function Manual Safety Integrated

4.4.6 Scratching/Determining the work offset

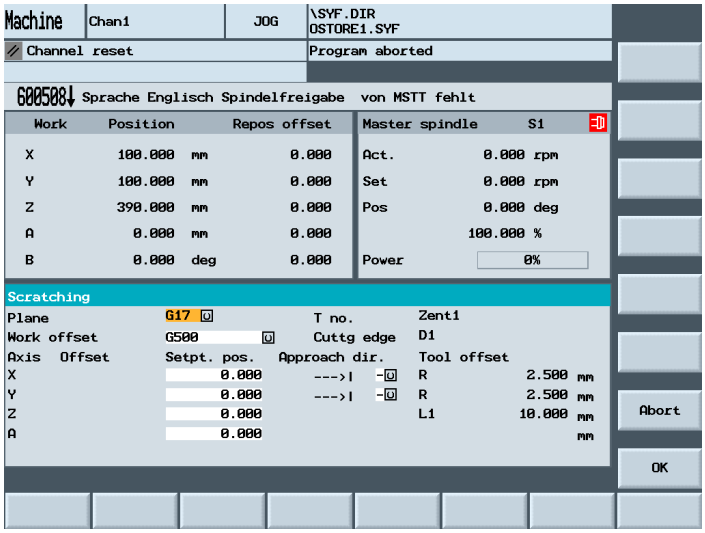
Function

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

Scratch

Sequence of operations

1. Press the "Scratch" softkey:
   - 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).



© Siemens AG 2008 All rights reserved.

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.
- "Position setpoint": Input of subsequent position setpoint for scratched edge.

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

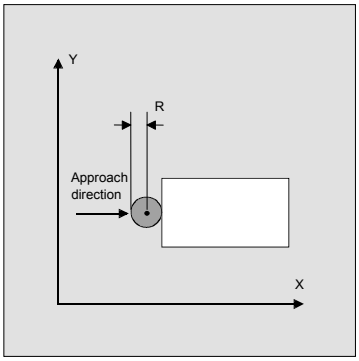
Use the cursor to select the first axis to be traversed in the "Scratch" display.

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 follow the machine manufacturer's instructions!

© Siemens AG 2008 All rights reserved.

Scratching in swiveled plane

Prerequisite:
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

Programming Manual Cycles: Swiveling

© Siemens AG 2008 All rights reserved.

4.5 MDA mode

4.5.1 Function and main screen

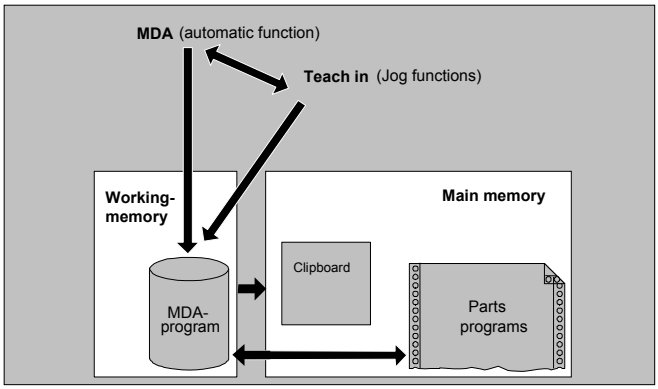
In "MDA" (Manual Data Automatic) mode, you can write part programs block by block and execute them. You can transfer the required motions as single part program blocks to the control using the operator panel.
The control starts processing the entered blocks when you press the "Cycle Start" key.

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.

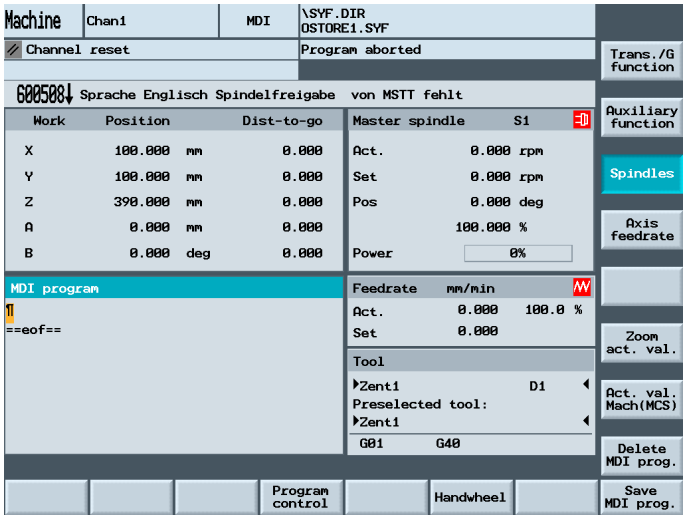
Teach In

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 executed 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 only be input and executed with the channel in the "Channel Reset" or "Channel interrupted" state.
The program created in MDA mode is saved in the "MPF" directory as a part program (MPF).

© Siemens AG 2008 All rights reserved.



The "MDA" main screen contains values relating to position, feedrate, spindle and tool as well as the contents of the MDA buffer.



Explanation of "MDA" basic display

As with the Jog main screen, the actual value window, spindle window, feedrate window and tool window are output.

Vertical softkeys

Like the Jog basic display, the MDA contains the "G Fct.+Transf.," "Auxiliary functions", "Spindles" (if spindle is configured), “Axis feed”, Zoom actual value”, and “MCS actual value” softkeys.

Delete MDA program

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

© Siemens AG 2008 All rights reserved.

4.5.2 Saving a program

Function

A program that has been created in MDA is saved in the "MPF"directory on the CompactFlash Card and inserted as a program.

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

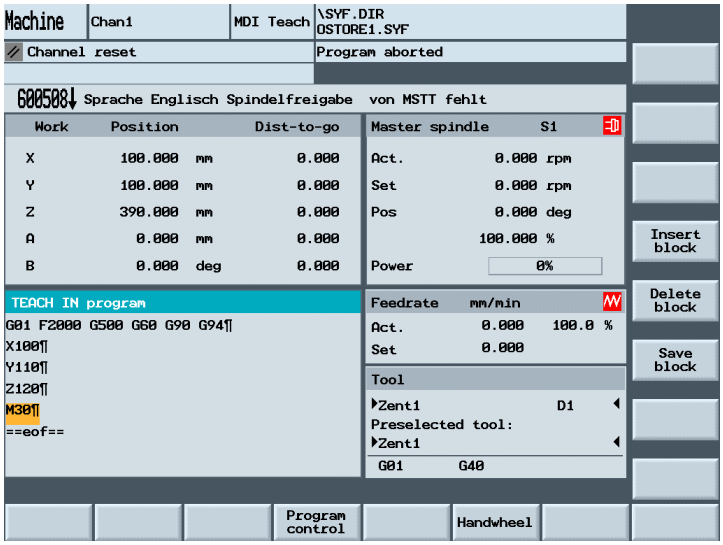
Function

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

There are two possible methods of writing programs with "Teach-in" and "MDI":

- Manual positioning
- Manual input of 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.



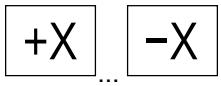
© Siemens AG 2008 All rights reserved.

1. Manual positioning

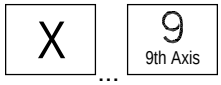
Sequence of operations

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

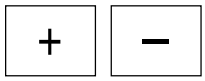
Submode "Teach-in" is selected.
After selection of the function, the cursor is located in the first blank line of the "Teach-in program" window.



Turning machine:
Press the "Axis" keys.



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 identifier and axis positions are transferred to the MDA program as values referred to the workpiece coordinate system.

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, auxiliary functions, etc.) in the program (if permitted).

© Siemens AG 2008 All rights reserved.

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.

Cycle Start

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 NCK are displayed in the "Current block" window.

2. Manual input of coordinates

Sequence of operations

"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.) to the program by entering them in the "MDA program" window.

Additional 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.

© Siemens AG 2008 All rights reserved.

4.6 Automatic mode

4.6.1 Function and main screen

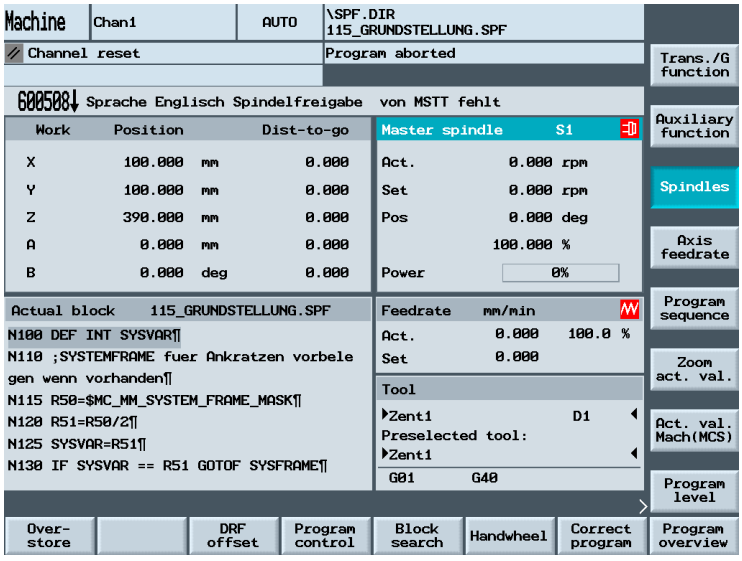
You can execute part programs fully automatically in "Automatic" mode; this is the normal operating mode for part machining.

Preconditions

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 downloaded the associated part program to 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" main screen contains values relating to position, feedrate, spindle and tool as well as the block currently being processed or program pointer.



Explanation of main screen

Like the Jog main screen, the Automatic main screen contains actual value, spindle, feedrate and tool windows.

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

© Siemens AG 2008 All rights reserved.

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 pointer" appears instead of the "Current block" window.
The program nesting depth (P = number of passes) is displayed.

Program level or Current block

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

Program blocks

7 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).

Additional information

The other softkeys are described in the sections below.

© Siemens AG 2008 All rights reserved.

4.6.2 Executing programs

Function

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

Sequence of operations

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 program in NCK

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 "Cycle Start" key, and you can track the axis position, spindle, etc in the display.

© Siemens AG 2008 All rights reserved.

4.6.3 Program editing

Function

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

Sequence of operations

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

It is not possible to overtravel the point of interruption. The feedrate override 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.

Close Cycle Start

After the override close the editor and continue processing 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!

© Siemens AG 2008 All rights reserved.

4.6.4 Setting the block search/search target

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 search with calculation, the same calculations are performed as in normal program control. The complete target block is then executed in the same way as with normal program execution.

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

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

You can define the search target by
- 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:
Function Manual, Fundamentals; Mode group, Channel, Program operation mode (K1)

Sequence of operations

"AUTO" mode is selected in the "Machine" operating area.
The channel is in reset status
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 target block.
The block search is started when you press one of the following three softkeys:

Calculate contour

Block search start with calculation on contour

© Siemens AG 2008 All rights reserved.

- 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 target 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 target in program editor:

Search position

The currently selected program level is displayed.

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

© Siemens AG 2008 All rights reserved.

Program level + or Program- level -

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

Search pointer

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

This contains input fields for program name, search type (block number, text, ...) and search target (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 target in the "Type" input field.

Information about the available search types is displayed in the user response line.

The following search types can be entered:

Search type (= jump to ...)	**Value in search type field**
End of program	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 target (according to search type) in the "Search target" field.

0 ... 9

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

Interrupt. point

Preassign search target as last program interruption point:
The search pointer is assigned the data of the last program interruption point.

© Siemens AG 2008 All rights reserved.

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 target 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 NCK.
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 NCK. 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 NCK, 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 overstore function to complete the necessary settings.

Sequence of operations

"AUTO" mode 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

Switches to the "Search position" dialog.

Search position

Define search target in program editor:
The currently selected program level is displayed.
The block search via "Search position" is only possible for programs which have been loaded onto the NCK and not for programs executed from the CompactFlash card.

© Siemens AG 2008 All rights reserved.

without calculation

Start search without calculation for external programs.

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

Program level + or Program-level -

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

Define search target in search pointer:

Search pointer

Once the "search pointer" softkey has been pressed, a screen form with the program pointer is displayed.
This contains input fields for program name, search type (block number, text, ...) and search target (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.

without calculation

Start search without calculation for 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 NCK. You can also enter and execute any NC block.

Sequence of operations

"AUTO" mode 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 NC blocks that are to be executed.

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.

© Siemens AG 2008 All rights reserved.

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

Note:

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

Additional 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.

© Siemens AG 2008 All rights reserved.

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)
DRF	Select DRF offset
PRT	Program test
SBL1	Single block with STOP after each machine function block
SBL2	Single block with STOP after each block

Display all blocks in the current block display.
Display only traversing blocks in the current block display.
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

Function Manual, Fundamentals; different interface signals (A2) or Function Manual, Fundamentals; Mode group, channel, program operation mode (K1)

Sequence of operations

"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.

OK

Confirm selection with "OK".

© Siemens AG 2008 All rights reserved.

4.6.8 DRF offset

Function

DRF (Differential Resolver Function) is a method of axial traversing that takes place in the basic coordinate system. Selecting the handwheel is only permissible via the axial interface; i.e. only machine axis names can be used.

Switch-on/switch-off

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 is performed for all axes
- DRFOF (selection of DRF via part program)
- PRESETON (modification of the actual value through Preset)

Alter DRF

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

Sequence of operations

"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 "DRF offset" window 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".

■

© Siemens AG 2008 All rights reserved.

© Siemens AG 2008 All rights reserved.

Parameters operating area

© Siemens AG 2008 All rights reserved.

© Siemens AG 2008 All rights reserved.

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 of 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.

© Siemens AG 2008 All rights reserved.

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 further entries, i.e., the tool type (drill, milling cutter, turning tools with cutting edge position, etc.).

Tool types

Tool type classification:

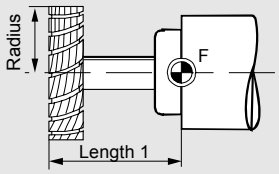
- **Group with type 1xy (milling cutter):**
  - 100 Milling cutter to CLDATA
  - 110 Ball-end cylindrical die-sinking cutter
  - 120 End milling cutter without corner rounding
  - 121 End mill with corner rounding
  - 130 Angle head cutter without corner rounding
  - 131 Angle head mill with corner rounding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

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

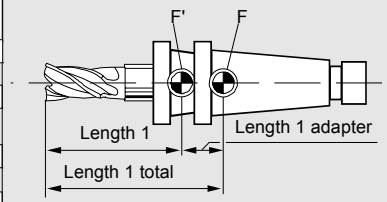
© Siemens AG 2008 All rights reserved.

Milling cutter with adapter

Entries in tool parameters	
DP1	1xy
DP3	Length 1 -Geometry
DP6	Radius -Geometry
DP21	Length -Adapter

Wear values according to requirements

Other values must be set to zero



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

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

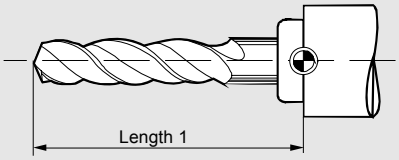
- **Group type 2xy (drills):**
  - 200 Twist drill
  - 205 Drill
  - 210 Boring bar
  - 220 Center drill
  - 230 Countersink
  - 231 Counterbore
  - 240 Tap regular thread
  - 241 Tap fine thread
  - 242 Tap Whitworth thread
  - 250 Reamer

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

© Siemens AG 2008 All rights reserved.

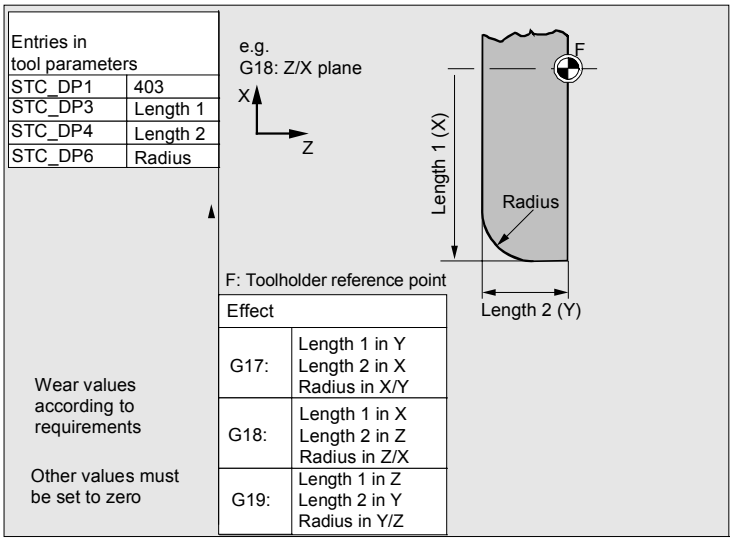
- **Group type 4xy (grinding tools)**:
  - 400 Surface grinding wheel
  - 401 Surface grinding wheel with monitoring
  - 403 Surface grinding wheel with monitoring without tool base dimension for grinding wheel peripheral speed GWPS
  - 410 Facing wheel
  - 411 Facing wheel with monitoring
  - 413 Facing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



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

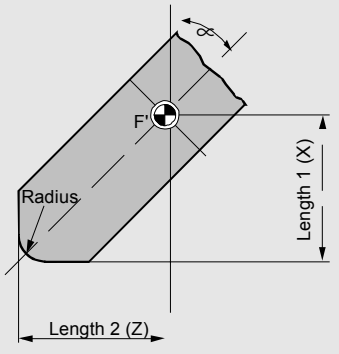
Wear values according to requirement

Other values must be set to zero

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



© Siemens AG 2008 All rights reserved.

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

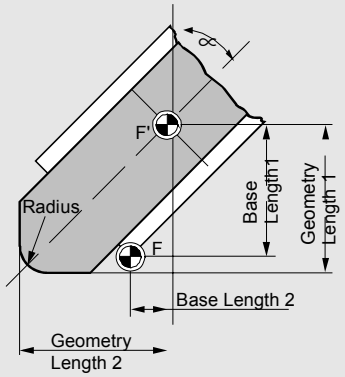
STC_TPG1	Spindle number
STC_TPG2	Chaining rule
STC_TPG3	Minimum wheel radius
STC_TPG4	Minimum wheel width
STC_TPG5	Current wheel width
STC_TPG6	Maximum speed
STC_TPG7	Max. surface speed
STC_TPG8	Angle of the inclined wheel
STC_TPG9	Parameter no. for radius calculation

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

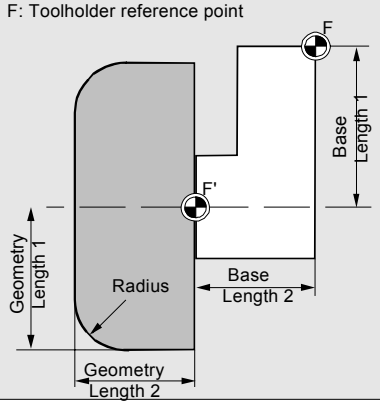
STC_TPG1	Spindle number
STC_TPG2	Chaining rule
STC_TPG3	Minimum wheel radius
STC_TPG4	Minimum wheel width
STC_TPG5	Current wheel width
STC_TPG6	Maximum speed
STC_TPG7	Max. surface speed
STC_TPG8	Angle of the inclined wheel
STC_TPG9	Parameter no. for radius calculation

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



© Siemens AG 2008 All rights reserved.

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

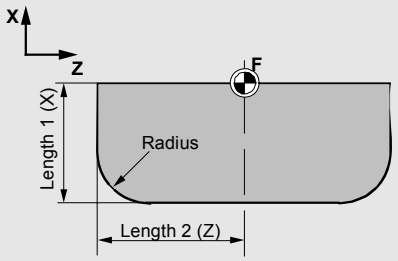
Other values must be set to zero

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



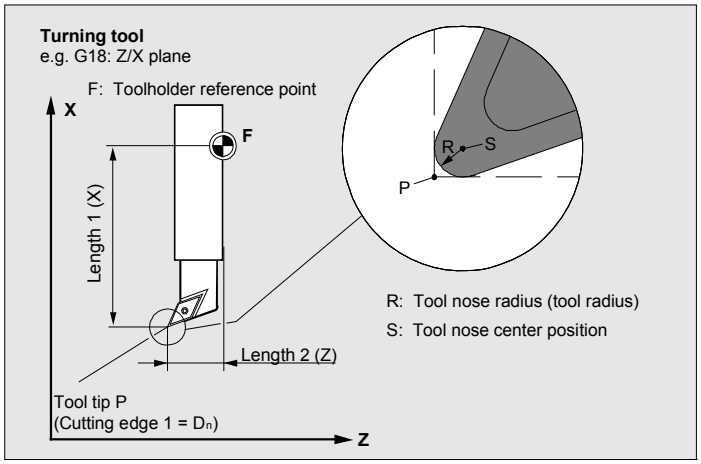
Assignment of tool-specific Parameter

Parameter	Description	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 peripheral speed	Real
$TC_TPG8	Angle of the inclined wheel	Real
$TC_TPG9	Parameter number for radius calculation	Integer
Additional parameters		
$TC_TPC1	Angle of the inclined wheel	Real
to		
$TC_TPC10		Real

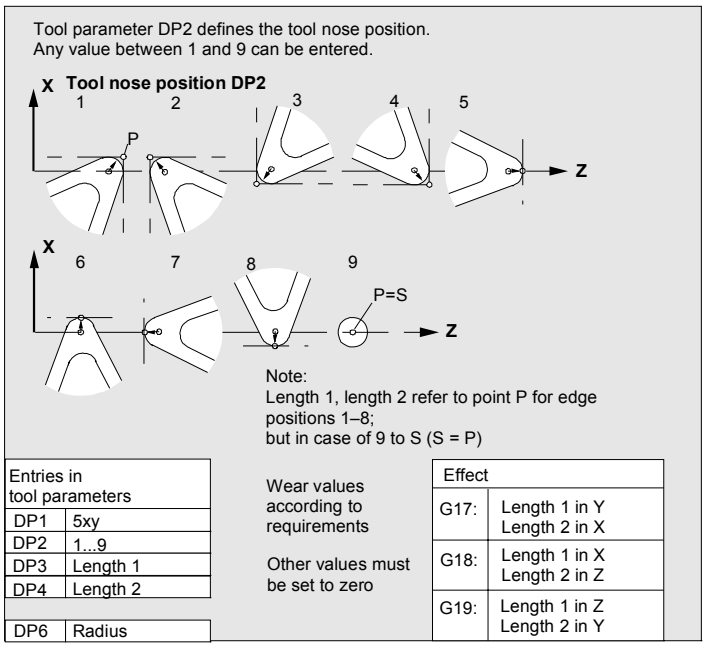
© Siemens AG 2008 All rights reserved.

- **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

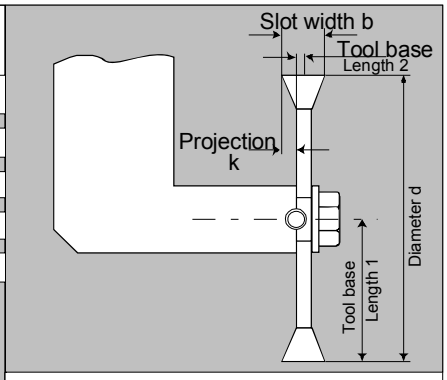
Effect	
G17:	Length 1 in Y Length 2 in X
G18:	Length 1 in X Length 2 in Z
G19:	Length 1 in Z Length 2 in Y

© Siemens AG 2008 All rights reserved.

- **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**	**Basis**	
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

© Siemens AG 2008 All rights reserved.

Calculation of 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)	Description	Comment
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	
Tool base dimension/ adapter	Length compensations	
21	Length 1	
22	Length 2	
23	Length 3	
Technology		
24	Clearance angle	For turning tools
25	Clearance angle	

© Siemens AG 2008 All rights reserved.

Offsets that are not required must be assigned the value 0 (= 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.

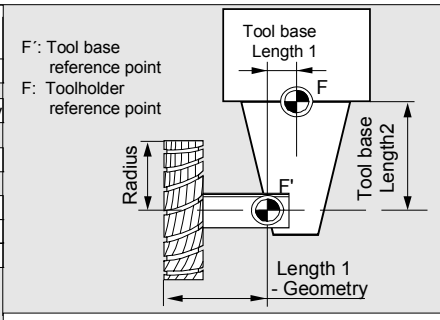
The tool offsets can be entered not only via the operator panel front but also via the data input interface.

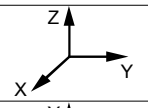
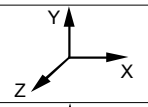
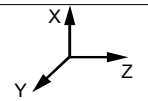
Calculation of tool base dimensions for two-dimensional millhead

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



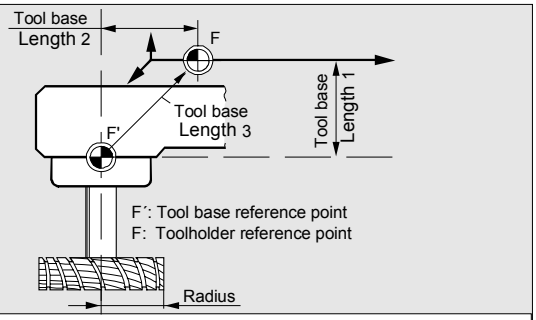
Effect		
G17:	Length 1 in Z Length 2 in Y Length 3 in X Radius in Y/Z	
G18:	Length1 in Y Length 2 in X Length 3 in Z Radius in X/Y	
G19:	Length 1 in X Length 2 in Z Length 3 in Y Radius in Z/X	

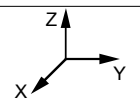
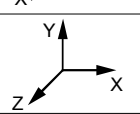
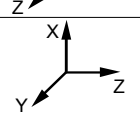
Calculation of tool base dimensions for three-dimensional millhead:

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

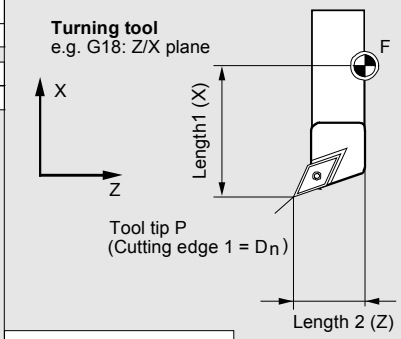


Effect		
G17:	Length 1 in Z Length 2 in Y Length 3 in X Radius in Y/Z	
G18:	Length 1 in Y Length 2 in X Length 3 in Z Radius in X/Y	
G19:	Length 1 in X Length2 in Z Length 3 in Y Radius in Z/X	

© Siemens AG 2008 All rights reserved.

Required length compensation values for turning tools:

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



Wear values according to requirements

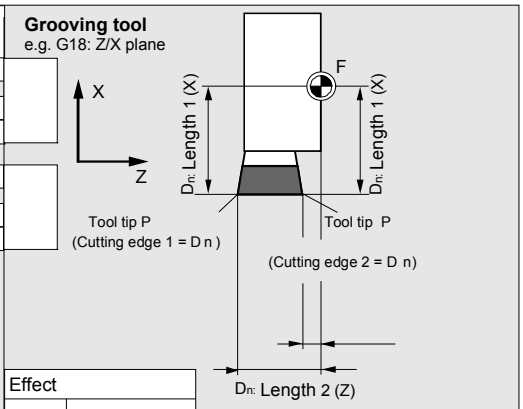
Other values must be set to zero

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

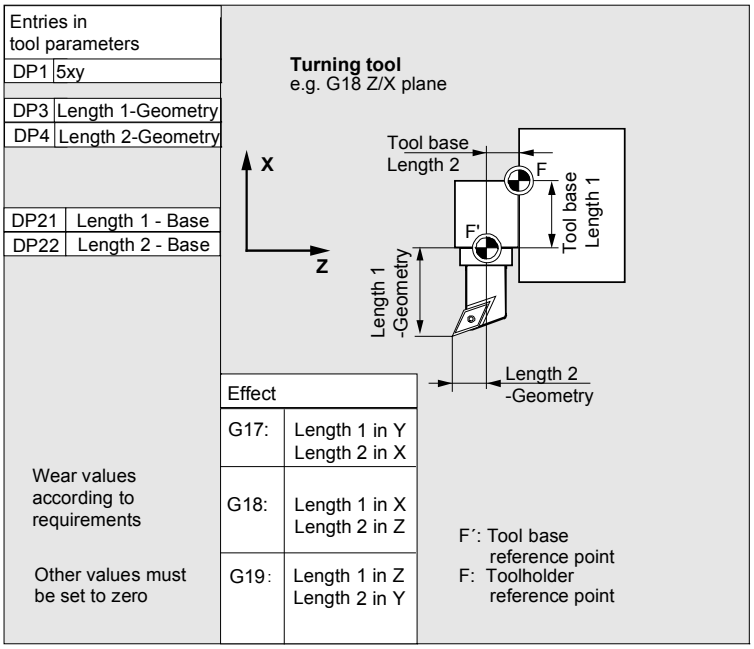
Other values must be set to zero

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

© Siemens AG 2008 All rights reserved.

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.

Parameter	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

© Siemens AG 2008 All rights reserved.

$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				
to				
$TC_DPC10				

* Value of the chaining parameter if the compensation parameters 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 peripheral 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 toolholder reference point and the tool tip in Cartesian coordinates.

© Siemens AG 2008 All rights reserved.

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

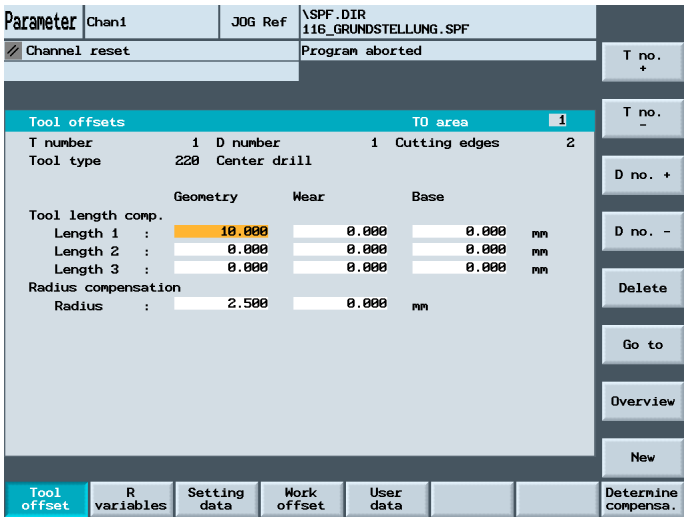
For programming of compensation data see
Function Manual Basic Functions

© Siemens AG 2008 All rights reserved.

5.2 Tool offset

5.2.1 Tool offset function and main screen

Tool offset data consist of data which describe 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!

© Siemens AG 2008 All rights reserved.

Horizontal softkeys

You can select different data types with the horizontal softkeys:

Tool offset	Selection of "Tool offset" menu
R Parameters	Selection of "R parameters" 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 highest offset number (cutting edge)
D No. -	Selection of next lowest 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

© Siemens AG 2008 All rights reserved.

5.2.2 Creating a 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.

Sequence of operations

The "Tool offset" window is displayed automatically.

New	Press the “New” softkey and then the "New tool" softkey.
New tool	

The "Create new tool" window is displayed and the classification of tool types appears beneath it.
1xx Milling tools
2xx Drilling tools
4xx Grinding tools
5xx Turning tools
7xx Special tools

As soon as you enter the first characters in the string for the tool type, 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

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

Abort	No new tool is created. Input is discarded.
OK	The new tool is created. The window is closed and the tool list is overlaid.

© Siemens AG 2008 All rights reserved.

5.2.3 Displaying a tool

Function

You can select tools that you have created and access their tool offset data.

Sequence of operations

The "Tool offset" window is displayed automatically.

If the "Parameters" 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 created tools.

Additional information

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

5.2.4 Finding a tool

Function

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

Sequence of operations

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

Overview

OK

Press the "Overview" softkey. The list of the available tools is displayed.
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.

© Siemens AG 2008 All rights reserved.

Go to

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

OK

2. Find tool by selecting softkey "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.

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

5.2.5 Deleting a tool

Function

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

Sequence of operations

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 the "Delete" softkey is selected.

Delete tool

Press the "Delete tool" softkey.
The tool with all edges is deleted and the tool offsets of the tool before the deleted tool are displayed.

© Siemens AG 2008 All rights reserved.

5.2.6 Creating a new cutting edge

Function

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

Sequence of operations

The "Tool offset" window is displayed automatically.

New | New edge

First press the “New” softkey and then the “New edge” softkey.
The "New cutting edge" window 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

See also section: Tool types and tool parameters

Abort

Press "Abort" to discard entered values.

OK

Press "OK" to save the entered values.

© Siemens AG 2008 All rights reserved.

5.2.7 Displaying a cutting edge

T No. + T No. -

Function

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

Select a tool of your choice and a cutting edge.

5.2.8 Finding a cutting edge

Overview

OK

Go to

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

OK

Function

You can use one of two search methods:

1. Find tools with edges by selecting the "Overview" softkey:
Press the "Overview" softkey. The list of the available tools is displayed.
Position the cursor on the tool you wish to find and confirm your request by pressing the "OK" softkey.

2. Find tools with edges by selecting the "Go to" softkey:
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.

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

5.2.9 Deleting a cutting edge

D No. + D No. -

Delete Delete edge

Function

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

Select the edge of a tool.

First press the "Delete" softkey and then the "Delete edge" softkey.

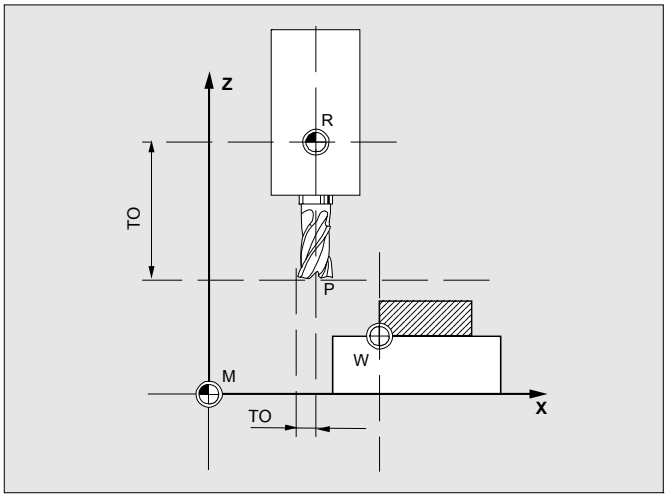
The displayed edge is deleted and the tool list updated.

© Siemens AG 2008 All rights reserved.

5.2.10 Determining tool offsets

Function

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



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

Sequence of operations

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

Determine compensa.

SELECT

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

OK

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

© Siemens AG 2008 All rights reserved.

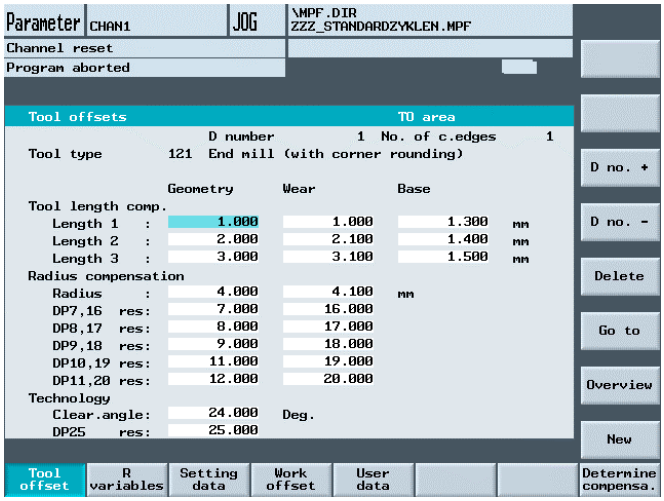
Calculate

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 Function Manual Basic Functions; Tool Offset (W1)

© Siemens AG 2008 All rights reserved.

5.2.12 Make active tool offset effective 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.

Additional 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 Function Manual, Basic Functions; Axes, Coordinate Systems, Frames (K2)

Caution

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

© Siemens AG 2008 All rights reserved.

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 search for, display and, in the majority of cases, edit 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 magazine location of each tool.

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

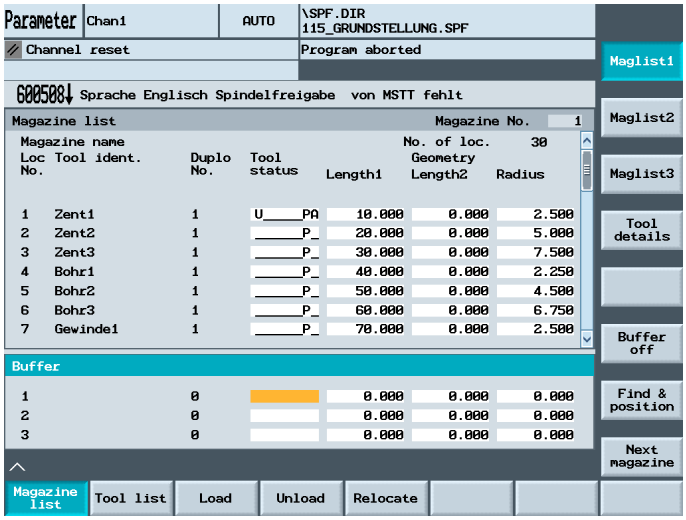
Commissioning Manual, Commissioning Base software and HMI-Embedded; Commissioning HMI-Embedded (IM2): Tool management
CNC Commissioning Manual: ShopMill; tool management
CNC Commissioning Manual: ShopTurn; tool management

© Siemens AG 2008 All rights reserved.

5.3.1 Tool management main screen

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
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

© Siemens AG 2008 All rights reserved.

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 NCK 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.

© Siemens AG 2008 All rights reserved.

Vertical softkeys

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

(Names assigned by user)

Maglist 1

Maglist 2

Maglist 3

- General data
- Geometry data
- Wear data

Tool details

Display and edit the tool offset data for a tool.

Buffer on or Buffer off

Display and hide 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 Selecting a 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".

© Siemens AG 2008 All rights reserved.

Sequence of operations

Tool mana-

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 Displaying and modifying 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:

- Corrective values
- Monitoring data

Sequence of operations

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.

© Siemens AG 2008 All rights reserved.

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)

displayed and modified.

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 Loading a 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).

© Siemens AG 2008 All rights reserved.

Sequence of operations

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. Finding an empty location
2. Current location
3. Load location

1.

Find empty location

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

2.

Current location

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

Load location

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

"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 entries with "OK".

© Siemens AG 2008 All rights reserved.

Tool list

Load from "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 a tool

Function

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

Sequence of operations

Magazine list

Unload from "Magazine 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.

© Siemens AG 2008 All rights reserved.

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

Tool list

Unload from "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.

Sequence of operations

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.

© Siemens AG 2008 All rights reserved.

or Find empty location

2. Select softkey "Find empty location" and an appropriate empty location is suggested.

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 a tool

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.

Sequence of operations

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.

© Siemens AG 2008 All rights reserved.

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 and the wear monitoring data in the tool list/tool wear list. You will be able to identify in 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 NCK, 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.

Loading/unloading
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 "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 (quantity)
- Monitoring of wear
- Additional tool status data (disable tool, tool in fixed location, oversized tool)

© Siemens AG 20086 All rights reserved.

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

Commissioning Manual, CNC Commissioning: ShopMill: Tool management

Operating Manual, Operating/Programming ShopMill: Tools and tool offsets

© Siemens AG 20086 All rights reserved.

5.4.1 Range of functions

Function

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

Tool types

– 120 End mill
– 200 Twist drill
– 220 Centering tool
– 710 3D probe
– 711 Edge probe
– 110 Cylindrical die-sinking cutter
– 111 Ballhead cutter
– 121 End mill with corner rounding
– 155 Bevel cutter
– 156 Bevel cutter with corner rounding
– 157 Tapered die-sinking cutter

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
Quantity
– Tool status: Tool disabled
– Tool status: Tool oversized (right and left half locations)
– Tool status: Tool in fixed location
– Fillet radius
– Angle for taper milling tools

Magazine parameters

– Magazine location disabled

5.4.2 Selecting a tool list

Sequence of operations

Tool list

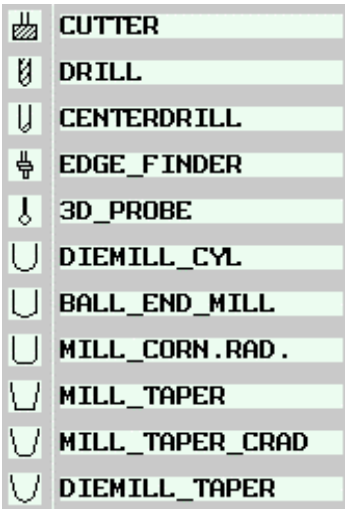
If “ShopMill” tool management is set up, the tool list menu is automatically displayed when you call up the “Parameters” operating area. Otherwise you can call the tool list via softkey.

© Siemens AG 20086 All rights reserved.

5.4.3 Creating a 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:



Sequence of operations

Attach the new tool to the spindle.

Tool list

Select the "Tool list" softkey. The tool list opens.

▲ ▼

Place the cursor on the location in the tool list that the tool occupies in the spindle. The location must still be vacant in the list.

New tool

Press the "New tool" softkey.

Cutter ... 3D tools

More

The vertical bar changes and you can select different tools via the softkey.
Additional tool types are available via the "More" softkey.

The new tool is created and automatically assumes the name of the selected tool type.

© Siemens AG 20086 All rights reserved.

Enter a unique tool name.

Enter the offset data of the tool.

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

Details

Press the "Details" softkey and enter the additional parameters.

The "Details" softkey is only active when a tool is selected for which additional information is required.

Name	Additional parameters
Angle head mill	Length2, Length3, ΔLength2, ΔLength3
Facing tool	Outside diameter, tool angle

3D tools

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

5.4.4 Creating more than one cutting edge per 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.

© Siemens AG 20086 All rights reserved.

Sequence of operations

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 "Cutting 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 highest or next lowest 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. "Facing tool_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 ("/").

© Siemens AG 20086 All rights reserved.

5.4.6 Creating a replacement tool

A replacement tool (also known as a duplo tool) is one that can be employed to perform the same machining operation as a tool that already exists (e.g. as a replacement after tool breakage).

When you create a replacement tool, you must use the same name as is used for a comparable tool.

Sequence of operations

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. 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!

© Siemens AG 20086 All rights reserved.

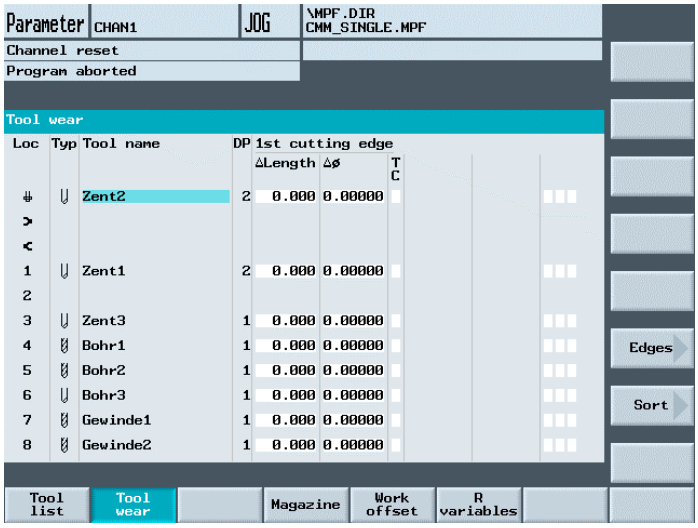
5.4.8 Entering tool wear data

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.
When you enter the wear data, ShopMill checks that the values do not exceed an incremental or absolute upper limit. The incremental upper limit indicates the maximum difference between the previous and new wear value. The absolute upper limit indicates the maximum total value that you can enter.
The upper limits are set in a machine data code.

Sequence of operations

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.

Machine manufacturer

Please note information supplied by machine tool manufacturer!

© Siemens AG 20086 All rights reserved.

5.4.9 Activating 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, identify them as oversize or assign them permanently to a magazine location. In the tool wear list you can assign each tool the following tool monitoring and properties:

- Tool life (T)
- Count (C)
- Wear (W)
- Other tool properties
  - Tool disabled (G)
  - Tool in fixed location (P)
  - Oversize tool (U)

The tool monitoring functions are activated via machine data.

Machine manufacturer

Please follow the machine manufacturer's instructions!

Sequence of operations

Tool wear

Select “Tool wear” via softkey

Tool life T (Time)

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.

© Siemens AG 20086 All rights reserved.

Count C (Count)	With the count, on the other hand, the number of times a tool is attached to a spindle is counted. The tool is also disabled in this case, when the remainder reaches "0".
Wear W (Wear)	With wear 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. The wear monitoring function must be set up by the machine manufacturer.
Machine manufacturer	Please follow the machine manufacturer's instructions!
Pre-warning limit	The pre-warning limit specifies a tool life, workpiece count or wear at which the first warning is displayed. The value for output of a warning due to the wear stage reached is calculated from the difference between the maximum wear and the warning limit entered.
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.)
Fixed-location-coded (F)	You can assign tools to a fixed location, i.e., the tool can only be used in its present magazine location. After machining, the tool always returns to its old magazine location.

Monitoring tool use

Tool wear	Select the "Tool wear" softkey
▲ ▼	Position the cursor on the tool that you want to monitor.

© Siemens AG 20086 All rights reserved.

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 the 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.

Entering tool statuses

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 assign the tool to a fixed magazine location.

The tool properties you have set become active immediately.

© Siemens AG 20086 All rights reserved.

5.4.10 Managing the magazine list

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.

Sequence of operations

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.

▲ ▼

Select the magazine location of your choice with the cursor keys.

Alternative

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 column "Tool status", you can see which properties have been assigned to the active tool:

- E: Tool is disabled
- U: Tool oversized
- P: Tool at a fixed location

Enabling a magazine location

▲ ▼

Position the cursor on the disabled field.

Alternative

Deselect option G in the "Location disable" column.
The magazine location is enabled again.

© Siemens AG 20086 All rights reserved.

5.4.11 Deleting a tool

Function

Tools can be deleted from the tool list.

Sequence of operations

Press the “Tool list” softkeys.

Select the tool of your choice with the cursor keys.

Press the "Delete tool" softkey and

confirm with "Delete".

The tool data of the selected tool are deleted, the magazine location in which the deleted tool was located is enabled.

The tool is not deleted with "Abort".

5.4.12 Changing a tool type

Function

In the tool list you can change a tool type into another tool type.

Sequence of operations

Press the "Tool list” softkey.

Select the desired tool and position the cursor on input field "Type."

Press the "Alternat." softkey until the tool type you are looking for appears.
The input fields for the new tool type are displayed.

© Siemens AG 20086 All rights reserved.

5.4.13 Loading or unloading a tool into or from the 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. Should 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.
Loading and unloading of tools into and out of magazine locations must be enabled in a machine data code.

Machine manufacturer

Please follow the machine manufacturer's instructions!

If your machine has only one magazine, you simply need to enter the location number you require when loading the tool, not the magazine number.

Sequence of operations

Loading a tool into the magazine

Tool list

Press the "Tool list" softkey.

▲ ▼

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 location number you require and press the "OK" softkey.
-or-

© Siemens AG 20086 All rights reserved.

Spindle

Press the "Spindle" and "OK" softkeys to load a tool into the spindle.

The tool is loaded into the specified magazine location.

Finding an empty location in the magazine and loading the tool

Magazine

Load

Select the "Magazine" softkey

Place the cursor on the tool that you want to load into the magazine.

Press the "Load" 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.

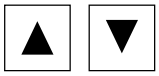
An empty location is suggested.

Press the "OK" softkey.

The tool is loaded into the suggested magazine location.

Unloading an individual tool from the magazine

Magazine



Unload

Select the "Magazine" softkey

Position the cursor on the tool that you want to unload.

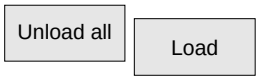
Press the "Unload" softkey.

The tool is unloaded from the magazine.

© Siemens AG 20086 All rights reserved.

Unloading 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 unloaded and then the process is aborted.
The unloading process is also aborted if you exit the magazine list.

5.4.14 Relocating a tool

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 that ShopMill will search for an empty location.

If your machine has just one magazine, you only 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 attach or detach a tool directly to or from the spindle.

Machine manufacturer Please follow the machine manufacturer's instructions!

© Siemens AG 20086 All rights reserved.

Specifying an empty location

Tool list

Press 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.

Finding 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.

© Siemens AG 20086 All rights reserved.

5.4.15 Positioning a magazine

You can position magazine locations directly on the loading point.

Positioning a magazine location

Magazine

Press the "Magazine″ softkey.

Place the cursor on the magazine location that you want to position on the loading point.

Position

Press the "Position" softkey.

The magazine location is positioned on the loading point.

5.4.16 Sort 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.

Sequence of operations

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 or By T number

Activate one of the softkeys to choose the sort criterion.

The tools are listed in the new order.

© Siemens AG 20086 All rights reserved.

5.5 ShopTurn tool management

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.

Loading/unloading

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

Operating Manual, Operating/Programming ShopTurn: Tool management
Commissioning Manual, CNC Commissioning: ShopTurn: Tool management

© Siemens AG 2008 All rights reserved.

5.5.1 Range of functions

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
- Threading tool
- Milling tool
- Drill
- Button
- Stop
- Threading tool
- Rotary drill
- 3D_probe
- Facing tool

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
- Number 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 locked

© Siemens AG 2008 All rights reserved.

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 cutter/drill) in diameter or radius via display machine data

5.5.2 Selecting a tool list

Sequence of operations

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.

5.5.3 Creating a 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.

© Siemens AG 2008 All rights reserved.

Sequence of operations

Install the new tool in the tool turret.

Tool list

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.

New tool

Press the "New tool" softkey.

Roughing tool ... 3D_Probe

More

Use the softkeys to select the tool type of your choice.
Additional tool types are available via the "More" softkey.

Edges

Select the required cutting edge position.

More

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 ("/").

If you assign a tool name that already exists, the "Create a duplo tool" window appears. You can decide whether you would like to create a sister tool (see Sec. “Creating duplo/replacement tools”.

Also enter the offset data of the tool.

▲ ▼

If you want to modify the cutting edge position of the tool later, place the cursor in the “Type” column.

Alternative or SELECT

Use the "Alternative" softkey or the "Select" key to select one of the specified options.

© Siemens AG 2008 All rights reserved.

Changing a tool name

You have the option of later changing a tool's name.

Position the cursor in the "Tool name" column and enter the desired name.

If you enter a tool name that already exists, the "Create a duplo tool" window appears. You are asked whether a duplo tool should be created.

Do not rename

Press the "Do not rename" softkey if you do not want to create a sister tool.

Enter a new tool name.

Abort

Using the "Abort" softkey, you can cancel the process at any time.

5.5.4 Creating more than one cutting edge per 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.

Sequence of operations

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 "Cutting 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 highest or next lowest edge number respectively.

© Siemens AG 2008 All rights reserved.

5.5.5 Creating replacement tools

A “replacement tool” (also known as a duplo tool) is a tool that can be used for the same machining operation as a tool that has already been input. 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.

For sister tools, the following data must agree with that of the original tool:

- Tool type
- Length of cutting edge
- Tool radius
- Direction of rotation
- Coolant

Sequence of operations

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.

Confirm the name with the "input" key and the duplo number of the replacement tool is automatically incremented by 1.

Now input the tool offset data of the tool.

The sequence for inserting a replacement tool is determined by the duplo number **DP**.

© Siemens AG 2008 All rights reserved.

5.5.6 Sorting 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.

Sequence of operations

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 Deleting tools

Tools that are no longer in use can be deleted from the tool list for a clearer overview.

Sequence of operations

Tool list

Press the "Tool list” softkey.

▲ ▼

Select the desired tool.

Delete tool

Press the "Delete tool" softkey and

✓ Delete

confirm with "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".

© Siemens AG 2008 All rights reserved.

5.5.8 Load or unload tool in the 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.

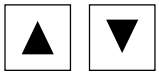
Loading and unloading of tool data into and out of magazine locations must be enabled in a machine data code.

Machine manufacturer

Please follow the machine manufacturer's instructions!

Sequence of operations

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.

The data relating to your tool are now displayed in the specified magazine location.

Unloading an individual tool from the magazine

Select the "Magazine" softkey

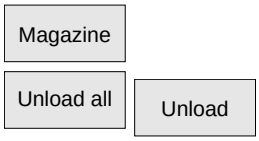
Position the cursor on the tool that you want to unload.

Press the "Unload" softkey.

The tool data are removed from the magazine and stored in the tool list in a position without a number.

© Siemens AG 2008 All rights reserved.

Unloading 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 unloaded and then the process is aborted.
The unloading process is also aborted if you exit the magazine list.

5.5.9 Relocating a tool

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 just one magazine, you only 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!

© Siemens AG 2008 All rights reserved.

Specifying 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.

Finding an empty location

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.

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.

© Siemens AG 2008 All rights reserved.

5.5.10 Positioning a magazine

You can position magazine locations directly on the loading point.

Positioning a magazine location

Magazine

Select the "Magazine" softkey

Position the cursor on the magazine location that you would like to position on the loading point.

Position

Press the "Position" softkey.

The magazine location is positioned on the loading point.

5.5.11 Entering 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.

Sequence of operations

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).

© Siemens AG 2008 All rights reserved.

5.5.12 Activating 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!

Sequence of operations

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!

Pre-warning 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.

© Siemens AG 2008 All rights reserved.

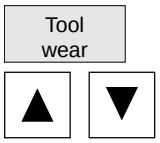
Oversize (U)

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.)

Fixed-location-coded (F)

You can assign tools to a fixed location, i.e., the tool can only be used in its present magazine location. After machining, the tool always returns to its old magazine location.

Monitoring tool use



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.
```

Entering tool statuses

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.

© Siemens AG 2008 All rights reserved.

5.5.13 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.

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.

Sequence of operations

Magazine

Press the "Magazine″ softkey.

Place the cursor on the relevant empty magazine location in the "Location disable" column.

Alternative

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.

© Siemens AG 2008 All rights reserved.

5.6 R parameters

5.6.1 Function

Function

Parameters are read and written by programs.
In this operating area, parameters can be edited manually.

5.6.2 Edit/delete/find R parameters

Function

The number of channel-specific R variables is defined in machine data.

Range:
R0 - R999 (dependent on machine data).
There are no gaps in the numbering within the range.

Sequence of operations

R Parameters

The "Channel-specific R parameters" window appears.
The channel-specific parameters are displayed.
The vertical softkey bar changes.

Editing parameters:
Position the cursor bar on the appropriate input field and enter the new values.

Deleting 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".

© Siemens AG 2008 All rights reserved.

Find

Finding parameters:
An input window for a parameter number appears when you press the "Find" softkey.

Enter the R parameter number you wish to find via the numeric keypad.

INPUT

When you press the "Input" key, the cursor is automatically positioned on this parameter if it exists.

Additional information

Input and deletion of parameters can be disabled via the keylock 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.

Sequence of operations

Setting data

Select softkey "Setting data".
The vertical softkey bar changes.

Working area

Press the "Working area limitation" softkey.
The "Working area limitation" window opens.

◀ ▶

Edit the 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.

© Siemens AG 2008 All rights reserved.

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.

Additional information

The "Working area limitation" function can be disabled by means of the keylock 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 Revolutional feedrate in mm (inch)/rev

Jog feedrate

Feedrate value in JOG mode

Continuous jog

- JOG mode: The axis moves as long as the key is pressed.
- Continuous mode: Axis moves after pressing key once, until
  - The key is pressed again,
  - NC Stop,
  - Reset,
  - Software/hardware limit switch.

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 leading spindle
- Direction of rotation: Direction of rotation of leading spindle
- Spindle speed: Speed of the master spindle in Jog Operation

© Siemens AG 2008 All rights reserved.

Sequence of operations

Setting data

Select softkey "Setting data".
The vertical softkey bar changes.

Jog data

Press the "Jog data" softkey.
The "Jog data" window is opened.

Edit JOG 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

Additional 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.

Sequence of operations

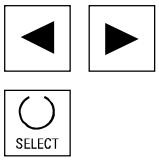
Setting data

Select softkey "Setting data".
The vertical softkey bar changes.

Spindle data

Press the "Spindle data" softkey.
The "Spindle data" window is opened.

© Siemens AG 2008 All rights reserved.



Edit 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.

Additional 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.

Sequence of operations

Setting data

Select softkey "Setting data".
The vertical softkey bar changes.

Feedrate DRY

Press the "Feedrate DRY" softkey.
The "Dry run feedrate" window is opened.

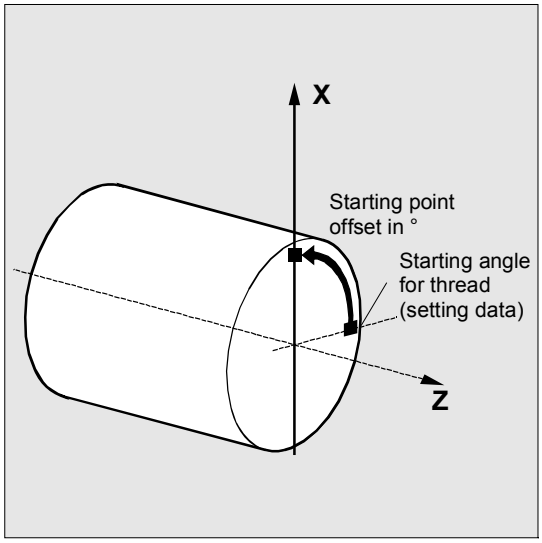
Edit the dry run feedrate:
Enter a new value.

© Siemens AG 2008 All rights reserved.

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.



Sequence of operations

Setting data

Select softkey "Setting data".
The vertical softkey bar changes.

Starting angle

Edit starting angle:
Press the "Starting angle" softkey.
The "Starting angle for thread" window opens.

Enter a new value.

© Siemens AG 2008 All rights reserved.

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.

Sequence of operations

Setting data

Select softkey "Setting data".
The vertical softkey bar changes.

Display setting data:

Misc.

Press the "Misc." softkey.
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).

Click "OK" to begin the find process.

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.

Additional information

Data can be edited or not depending on the active access protection level.

© Siemens AG 2008 All rights reserved.

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 Programming Manual Work Preparation

Sequence of operations

Setting data

Select softkey "Setting data".
The vertical softkey bar changes.

Protection zones

Press the "Protection zones" softkey.
The "Working area limitations and protection zones" window opens.
The vertical softkey bar changes again.

Protection zone + | Protection zone -

Press the "Protection zone +" or "Protection zone -" softkey.
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; infeed direction Z)

G18

- Plane G18 (Z, X; infeed direction Y)

G19

- Plane G19 (Y,Z; infeed direction X)

© Siemens AG 2008 All rights reserved.

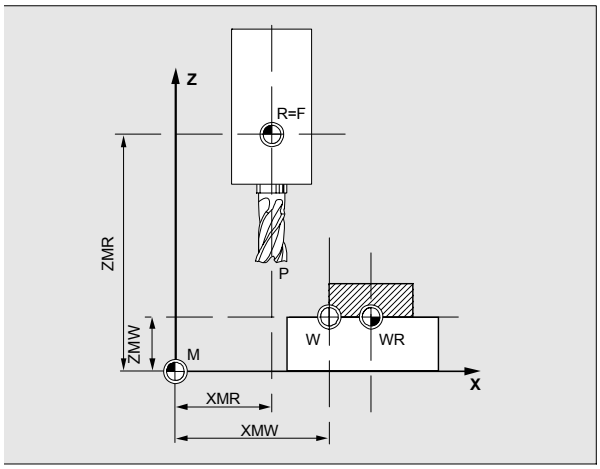
5.8 Work offset

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	Work offset
M	Machine zero
R	Machine reference point
WR	Workpiece reference point

Effective WO

The work offset effective in an axis $P_ACTFRAME=.. results from the **sum** of the following zero point offsets:

Settable WO

You can activate a settable zero point offset in the part program you have called with G54 to G57 and other G functions or with $P_IFRAME=..
Basic work offset (basic frame): displayed like a settable WO.

© Siemens AG 2008 All rights reserved.

Programmable WO

You can use the programmable zero point offset $P_PFRAME=.. to program an additional zero point 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, an external work offset can be overlaid by means of the handwheel (DRF offset) or from the PLC.

DRF offset

Differential Resolver Function: NC function which generates an incremental work 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

Programming Manual Work Preparation

Frame components

Frame components
A frame can consist of the following arithmetic rules:

- Work 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.

© Siemens AG 2008 All rights reserved.

5.8.2 Displaying work offsets

Function

In the overview, all existing settable work offsets are listed. The number of possible work offsets is defined by a machine data.
The first settable work offsets G54 to G57 are permanently assigned the identifiers `$P_UIFR[1]` to `$P_UIFR[4]`.

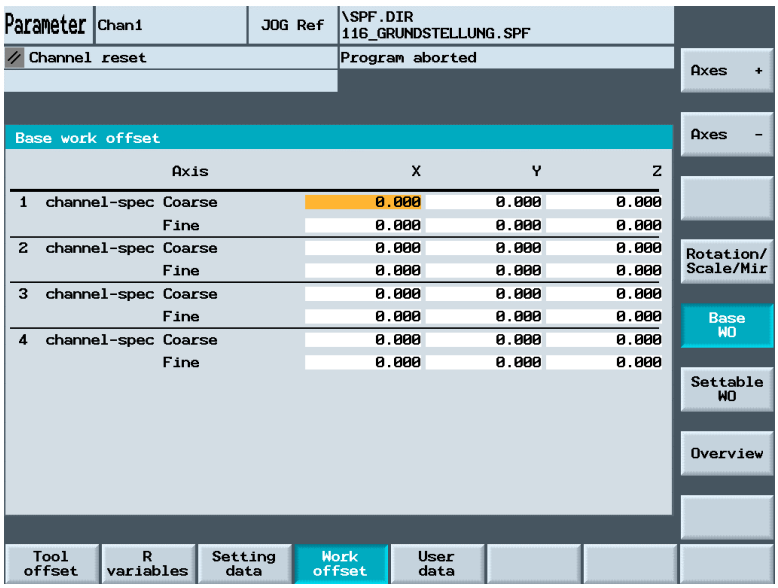
Sequence of operations

Work 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.

© Siemens AG 2008 All rights reserved.

Offsets | Rotation scal., mirr.

You can use these softkeys to change the display mode of the currently displayed work offsets.
The following takes place:

- 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, edit 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 WO

All defined settable work offsets are displayed in a table where they can be edited if necessary (select and edit).

5.8.3 Changing settable work offset (G54 ...)

Function

$P_UIFR[] This identifier can be used to edit a settable work 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.

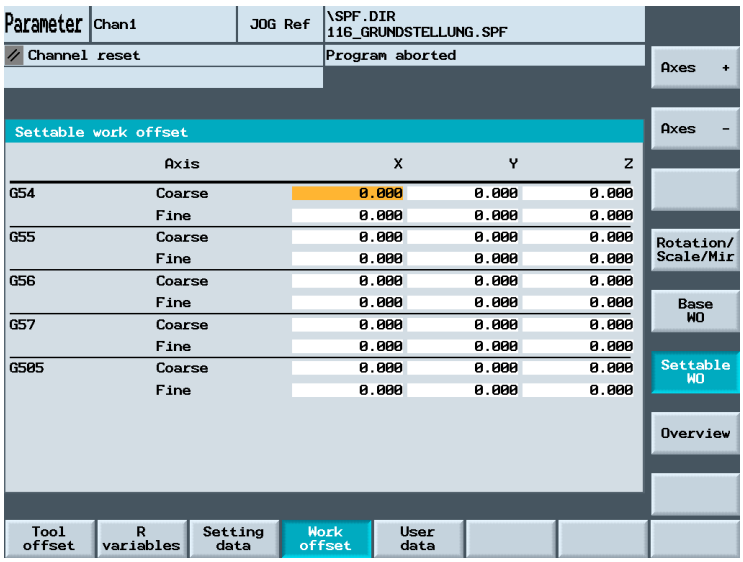
© Siemens AG 2008 All rights reserved.

Sequence of operations

Work offset

Settable WO

Select softkey "Work Offset".
The vertical softkey bar changes.
The "Settable work offsets" window opens.



▲ ▼

SELECT

INPUT

You can choose a work offset selectively from the overview of work offsets. Use the cursor to select the corresponding fields and overwrite the fields to be changed with new values.
Select a new value via the "Select" key (with mirroring).

The zero point offsets are saved, i.e. transferred to the NCK.

© Siemens AG 2008 All rights reserved.

5.8.4 Activate work 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.

Additional 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 Function Manual, Basic Functions; Axes, Coordinate Systems, Frames (K2)

Danger

The offset is applied the next time the part program is started.

5.8.5 Global work 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 TO (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).

© Siemens AG 2008 All rights reserved.

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.
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

Function Manual, Basic Functions; Axes, Coordinate Systems, Frames (K2)

Sequence of operations

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.

Additional 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.

© Siemens AG 2008 All rights reserved.

5.9 Displaying system frames

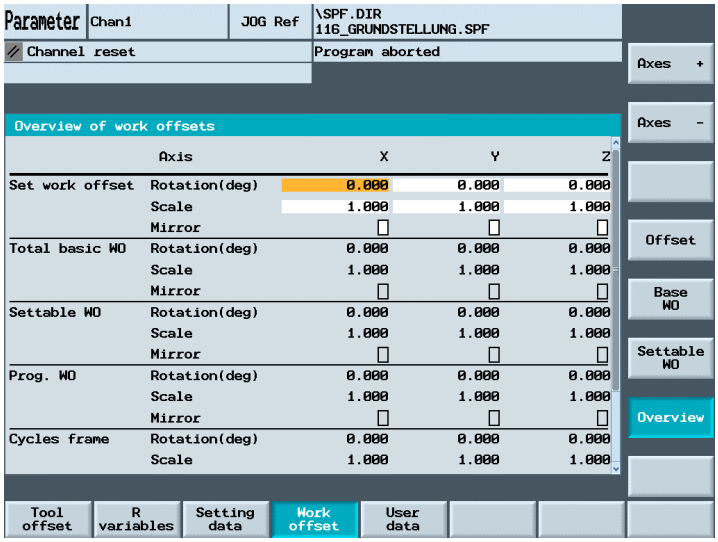
If system frames are activated via MD, they can be displayed via the “Parameter” operating area, Work offsets.

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 follow the machine manufacturer's instructions!

© Siemens AG 2008 All rights reserved.

5.10 User data/user variables (GUD, PUD, LUD)

5.10.1 General information

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 subprogram 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.

5.10.2 Change/find user data/user variables

Sequence of operations

User data

Press the "User data" softkey.
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-specific UD

- "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.

© Siemens AG 2008 All rights reserved.

Searching for user data

GUD + GUD -

Press the "GUD +" and "GUD - " softkeys 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)
- 4 ... 9 = GD4… GD9 (additional, e.g., grinding cycles, etc.)

The selected data are displayed in the "Global user data" window.

Find

Press the "Find" softkey.
The "Find user data" dialog window 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.

Additional information

How to define and activate user data is described in
Section 6: Program operating Area.

■

© Siemens AG 2008 All rights reserved.

Program operating area

© Siemens AG 2008 All rights reserved.

6.1 Program types

6.1.1 Part program

A part program consists of a sequence of instructions to the NCK 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 type 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.

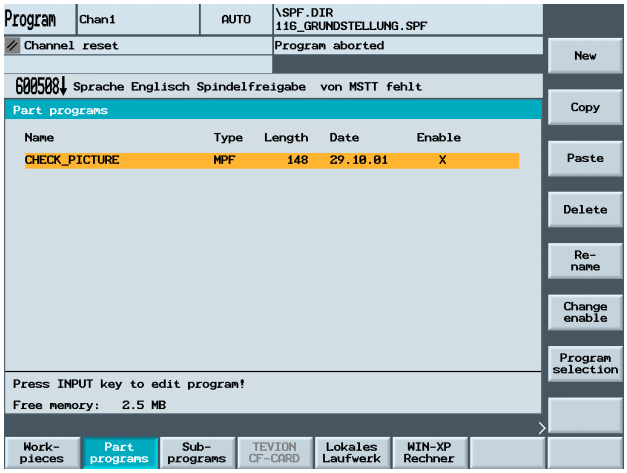
6.1.5 Storing programs

Programs are saved in the NCK memory. The size of this memory is dependent on settings made during start-up. (See Section 6 "Memory info")

© Siemens AG 2008 All rights reserved.

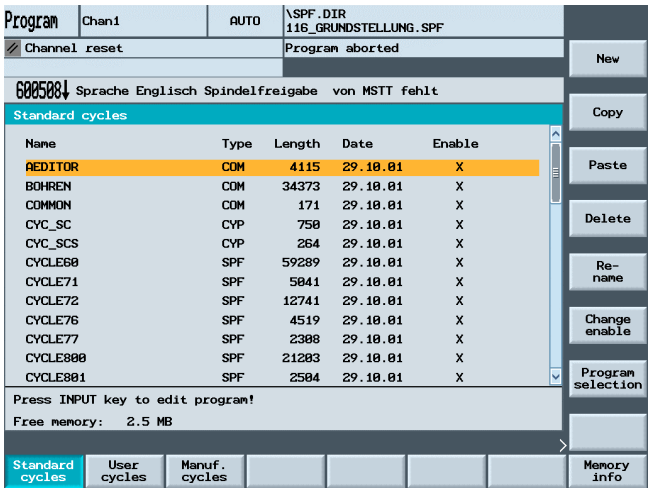
6.2 Program main screen

The program's main screen contains a complete overview of all workpiece and program directories.



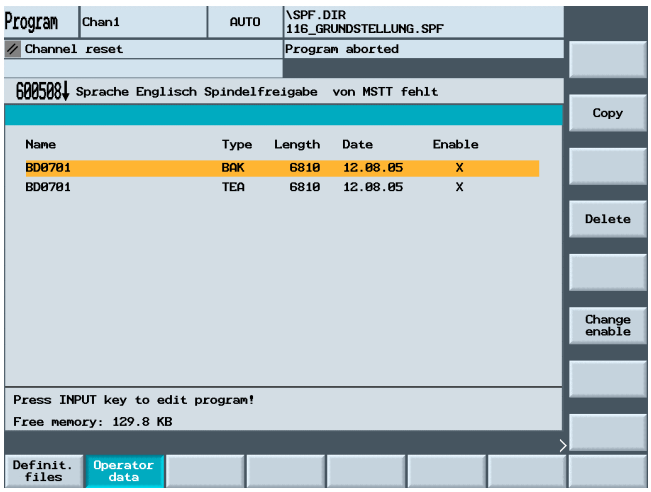
>

Press the "Etc." key to open the following screen:



>

Press the "Etc." key to open the following screen:



© Siemens AG 2008 All rights reserved.

Horizontal softkeys

Work-pieces
This key displays an overview of all the workpieces you have created with their part programs.

Part program
An overview of all part programs (main programs) stored in the selected directory is displayed.

Sub-routines
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".

TEVISON-CF-CARD, Local drive, WIN-XP computer
You can configure as many as eight connections (logical drives). Four softkeys e.g. "TEVISON CF-CARD, "local drive", "WIN-XP computer", appear on the first window level.

>
You can access additional (up to four) configured drives via the ETC key.

Standard cycles
You can use the "standard cycles" softkey to list the standard cycles.

User cycles
Select the "User cycles" softkey to display a list of the cycles that you have added.

Manufact. cycles
Select the "Manufacturer cycles" softkey to display a list of the cycles that the manufacturer has added.

Memory info
The total available/used memory is displayed.

>
You can access the following additional softkeys by pressing the ETC key.

Definition files
You will receive a list of the definition files (e.g. GUD4_DEF)

Operating data
You will receive the operation data files (e.g. BD_TEA).

Machine manufacturer
These softkeys can be disabled in display machine data.
Please note information supplied by machine tool manufacturer!

© Siemens AG 2008 All rights reserved.

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.
Paste	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.
Memory info	The current memory of the NCK main memory is displayed.
Activate	You can activate the definitions of a file with the “Activate” softkey. Confirm or reject the activation with the “yes” or “no” softkeys.

6.3 Editing programs

6.3.1 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".

© Siemens AG 2008 All rights reserved.

6.3.2 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."

Additional information

When creating a program template, please make sure that the read-only identifier appears immediately at the end of the block.

© Siemens AG 2008 All rights reserved.

6.3.3 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.

Settings

Machine manufacturer

This function must be set up by the machine manufacturer. Please note the information supplied by the machine tool manufacturer!

6.3.4 Reserved character string

Function

Reserved character strings occur in part program code lines that arise by calling cycles and contour programming. They can be viewed by setting "Display hidden lines" in the editor.

The following character strings must not be used in part program lines that are input directly:
;#
;#END
;NCG
;*RO*
;*HD*

Additional information

See Section: Selective Program Protection RO
Hidden program lines: display HD

© Siemens AG 2008 All rights reserved.

6.3.5 Define and activate 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

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?"

No

"None" 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

"None" The screen form is closed, the definitions are not activated

© Siemens AG 2008 All rights reserved.

Error messages:
The following error messages appear in the dialog line:
"Error occurred during activation of file"

Output of NCK alarms:
Once the maximum number of files on the NCK has been reached, no more backup files can be created. The following acknowledgeable NCK alarms appear: “Too many part programs in the NC memory”
"NC memory full"

The same applies when activating macro files (.MAC).

© Siemens AG 2008 All rights reserved.

6.4 Free contour programming

6.4.1 General information

Function

The free contour programming is a support tool for the editor.
The contour programming function enables you to create simple and 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:

- Straight vertical line

- Straight line (planar, longitudinal, inclined) horizontal

- Diagonal line

- Arc/circle

Additional 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

© Siemens AG 2008 All rights reserved.

6.4.2 Graphical illustration 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	Description
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 contour is displayed to the extent 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 must be entered. Check the contour elements you have already programmed, if required. You may have forgotten to enter all of the known data.

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.

© Siemens AG 2008 All rights reserved.

6.4.3 Creating a 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.

Process

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:

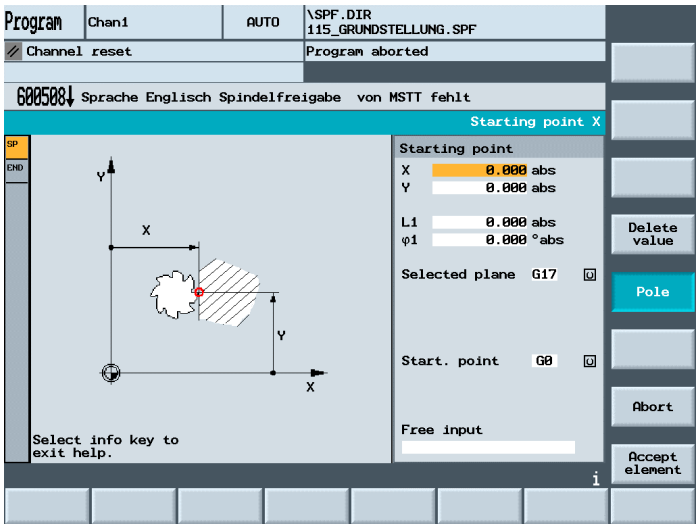
Support | New Contour

Open the contour editor by pressing the “Support" and "New contour" softkeys.

© Siemens AG 2008 All rights reserved.

Defining the start point

The input screen for the start point of the contour appears.



When entering a contour, begin at a position which you already 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 starting point axes are automatically adjusted.

Alter-native

Position the cursor on the "Transverse axis dimension" field and click on the field using the "Alternative" softkey (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 the "Accept element" softkey to store the starting point.

Abort

With the “Abort” softkey, your settings are discarded and you arrive at the previous screen form.

© Siemens AG 2008 All rights reserved.

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 Sec. "Creating 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 Sec. "Creating contour elements").

Close 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 "Recompile" softkey. The editor cursor must be positioned inside the contour to do this.

Notice

Upon recompilation only the contour elements that were created with free contour programming are regenerated. In addition, only the texts that were added using the "Free text input" input field are recompiled. Any changes you made directly in the program text are lost. However, you can subsequently insert and edit user-defined texts, which will not be lost.

© Siemens AG 2008 All rights reserved.

Additional 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

6.4.4 Changing a contour

You can change a previously created contour later.
Individual contour elements can be

- appended,
- modified,
- added
- 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.

Process

Modifying contour element

Select the contour.

Press the "Cursor Right" key.

Position the cursor on the contour element that you want to modify.

Press the “INPUT” key

The associated input form is opened and an enlarged view of the selected element appears in the programming graphics.

Accept element

After inputting the changes, press the "Accept element" softkey.

© Siemens AG 2008 All rights reserved.

6.4.5 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	Icon	Description
Starting 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
top/bottom	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	Description
-	Black	Cursor on new element
White	Black	Cursor on current element
Black	White	Normal (undefined) element
White	Black	Element currently detached (residual model)

© Siemens AG 2008 All rights reserved.

6.4.6 Create, change, delete contour elements

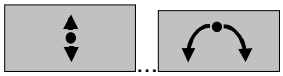
When you have created a new contour and specified the start point, you can define the individual elements that the contour comprises.

If you leave any parameter input fields blank, the control assumes that you do not know the right values and attempts to calculate these from the settings of the 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.

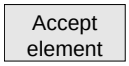
You can select an existing contour element with "INPUT". A new contour element is inserted after the cursor when you select one of the contour elements on the horizontal softkey menu; the input focus is then switched to the parameter input on the right of the graphic display. You can navigate around 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.

Select contour element

Position the cursor on the desired contour element in the contour chain, and select using 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.

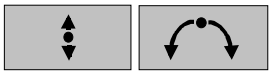
© Siemens AG 2008 All rights reserved.

Append contour element

Select the contour.

Press the "Cursor Right" key.

Place the cursor on the last element before the end of the contour.



Select the required contour element via softkey.

Enter the parameters in the input screen.

Accept element

Press the "Accept element" softkey.

The required contour element is appended to the contour.

Displaying additional parameters

If your drawing contains further data (dimensions) for a contour element, select the "All parameters" softkey to extend the range of input options for the element.

All parameters

Alter-native

The "Alternative" softkey is displayed only in cases where the cursor is positioned on an input field with several switchover settings.

Defining a pole

If you wish to enter the contour elements Diagonal line and Circle/arc in polar coordinates, you must first define a pole.

Continue Pole

Press the "Continue" and "Pole" softkeys.

Enter the coordinates of the pole.

Accept element

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.

© Siemens AG 2008 All rights reserved.

Tangent to preceding element

When entering data for a contour element you can program the transition to the preceding element as a tangent.

Tangent to prec. Ele

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.

Select dialog

Some parameter configurations can produce several different contour characteristics. In such cases, you will be asked to select a dialog.

Select dialog

Press the "Select dialog" softkey to switch between the two different contour options.
The selected contour appears in the graphics window as a solid black line and the alternative contour appears as a dashed green line.

Accept dialog

Press the "Accept dialog" softkey to accept the chosen alternative.

Changing dialog selection

If you want to change an existing dialog selection, you must select the contour element in which the dialog was originally chosen.
Open the input screen form for the contour element.

Change selection

Press the "Change selection" softkey.

The two selection options appear again.

Select dialog

Press the "Select dialog" softkey to switch between the two different contour options.

Accept dialog

Press the "Accept dialog" softkey.

The chosen alternative is accepted.

Transition element at contour end

A transition element can be used whenever there is a point of intersection between two neighboring elements; this can be calculated from the input values.
You can choose among Radius **R**, a chamfer **FS**, and an **undercut** (thread, thre. 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 for the relevant contour element.

© Siemens AG 2008 All rights reserved.

Place the cursor on the last contour element.
Press the "Right cursor" 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 contour element

Select the contour.

Press the "Right cursor" 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.

Save a contour 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 main screen.
You can then program the next contour element.

Accept element

Additional 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.

© Siemens AG 2008 All rights reserved.

Parameters on gray background

These parameters have been calculated by the control and cannot be altered.
When the programmable parameter input fields (white background) are altered, the control calculates new data, which are then immediately displayed in the input screen.

Input value is already calculated

With some contours, the control 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 precise value 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".

6.4.7 Help

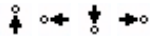
When you input parameters, you can call up a help screen using the Info key which graphically represents the parameters you are entering. The help screen that appears depends on the cursor position in the parameter display.
The help screen is displayed on top of the parameter screen.

If you press the Info key again the help screen is closed and the graphic display is activated once 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:

- Starting point
- Straight vertical line
- Straight vertical line, angle entry field
- Straight horizontal line
- Straight horizontal line, angle entry field
- Straight line in any direction
- Straight line in any direction, angle entry field
- Circle
- Circle, angle entry field
- Radius/chamfer

© Siemens AG 2008 All rights reserved.

6.4.8 Parameter description of straight line, circle and pole contour elements

Parameter	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 straight line	mm
$\alpha 1$	Pitch angle with reference to X axis	Degrees
$\alpha 2$	Angle to preceding element; tangential transition: $\alpha 2=0$	Degrees
FB	Feedrate for contour element "Straight line"	mm/rev
Transition to 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	

© Siemens AG 2008 All rights reserved.

Parameter	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
$\alpha 1$	Starting angle with reference to X axis	Degrees
$\alpha 2$	Angle to preceding element; tangential transition: $\alpha 2=0$	Degrees
$\beta 1$	End angle with reference to X axis	Degrees
$\beta 2$	Angle of aperture of circle	Degrees
FB	Feedrate for circle contour element	mm/rev
R	Radius of circle	mm
I	Position of circle 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 circle center point in Y direction (abs. or incr.)	mm
Transition to next 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.

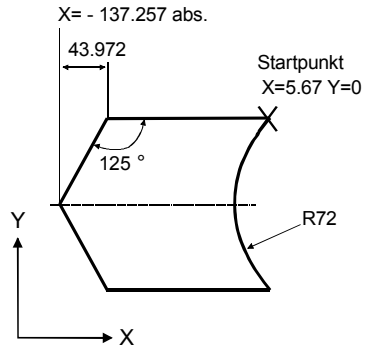
© Siemens AG 2008 All rights reserved.

6.4.9 Programming examples for 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



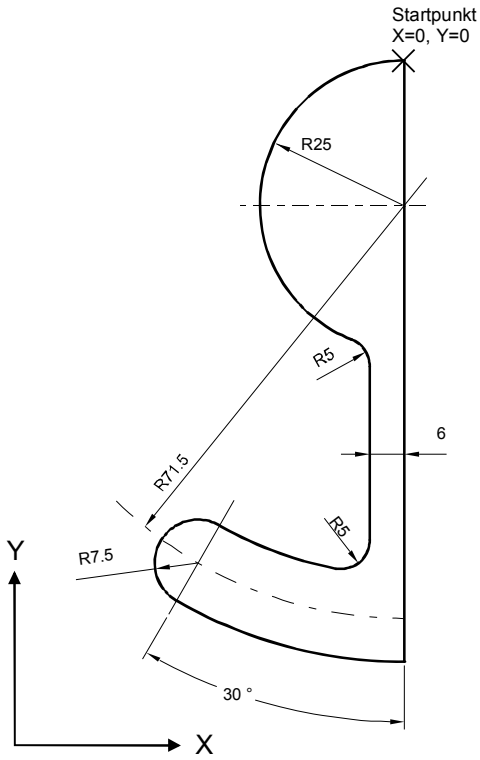
Element	Softkey	Parameter	Comment
1		All parameters, α1=180 degrees	Observe angles in help screen!
2		X=–43.972 inc, all parameters X=–137.257 abs. α1=–125 degree	Definition of coordinates in X in "abs" and in "inc" Observe angles in help screen!
3		X=43.972 inc α1=–55 degree	Definition of coordinates in X in "inc" Observe angles in help screen!
4		X=5.67 abs	
5		Clockwise direction of rotation, R=72, X=5.67 abs.., Y=0 abs.., Select dialog	

© Siemens AG 2008 All rights reserved.

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 "All parameters" softkey.

Workpiece drawing of contour



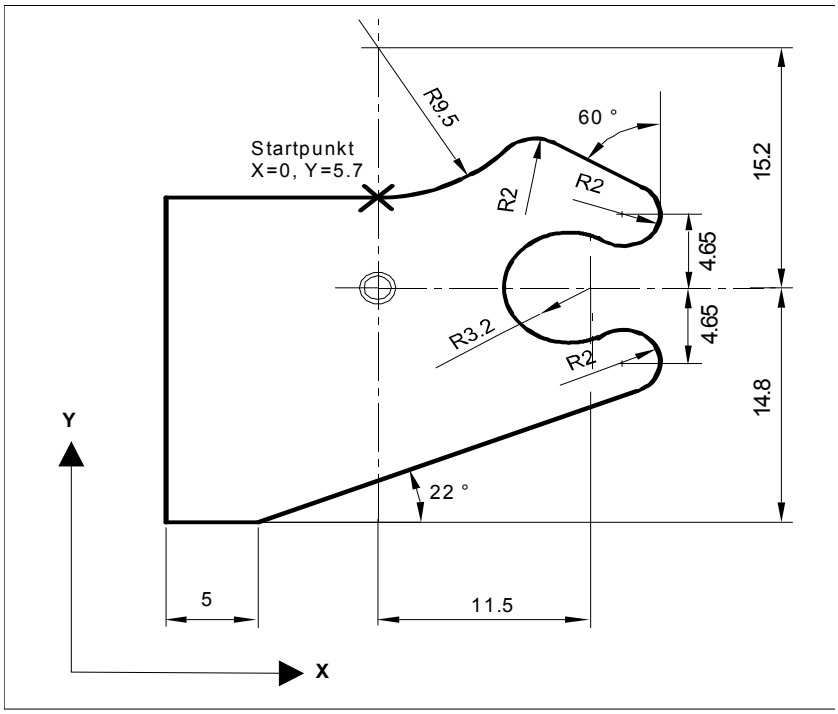
Element	Softkey	Parameter	Comment
1		Y=–104 abs.	
2		Clockwise direction of rotation, R=79, I=0 abs., Select dialog, all parameters, β2=30 degrees	
3		Clockwise direction of rotation, tangent to preced. R=7.5, all parameters, β2=180 degrees	
4		Counterclockwise direction of rotation, R=64, X=–6 abs., 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		Clockwise direction of rotation, R=25, X=0 abs., Y=0 abs. I=0 abs., make dialog selection, make dialog selection	

© Siemens AG 2008 All rights reserved.

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	Parameter	Comment
1		Counterclockwise direction of rotation, R=9.5, I=0 abs., make dialog selection, Transition to following element: R=2	
2		α1=–30 degree	Observe angles in help screen!
3		Clockwise direction of rotation, tangent to preced. R=2, J=4.65 abs.	
4		Counterclockwise direction of rotation, tangent to preced. R=3.2, I=11.5 abs., J=0 abs., make dialog selection, make dialog selection	
5		Clockwise direction of rotation, tangent to preced. R=2, J=–4.65 abs., make dialog selection	
6		Tangent to preced. α1=–158 degrees, Y=–14.8 abs., α2=0 degrees	Observe angles in help screen!
7		All parameters, L=5, make dialog selection	
8		Y=5.7 abs.	
9		X=0 abs.	

© Siemens AG 2008 All rights reserved.

6.5 Program simulation

6.5.1 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

Function Manual, Basic Functions; Axes, Coordinate System, Frames (K1): Mode group, program operation mode

© Siemens AG 2008 All rights reserved.

Sequence of operations

Simulation

Select a program, open it, and press the "Simulation" softkey.

Cycle Start

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 coarse or Cursor fine

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.

© Siemens AG 2008 All rights reserved.

6.5.2 Simulating milling 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

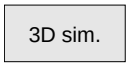
This function is an option and is only available with a color display. Milling simulation is only possible in the 1st channel.

Sequence of operations

Requirements

- 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. font.
- 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.

© Siemens AG 2008 All rights reserved.

6.5.3 Simulating milling 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.

Sequence of operations

Requirements

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>")
```


© Siemens AG 2008 All rights reserved.

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:
Operating Manual, Operating/Programming ShopMill

© Siemens AG 2008 All rights reserved.

6.6 Managing programs

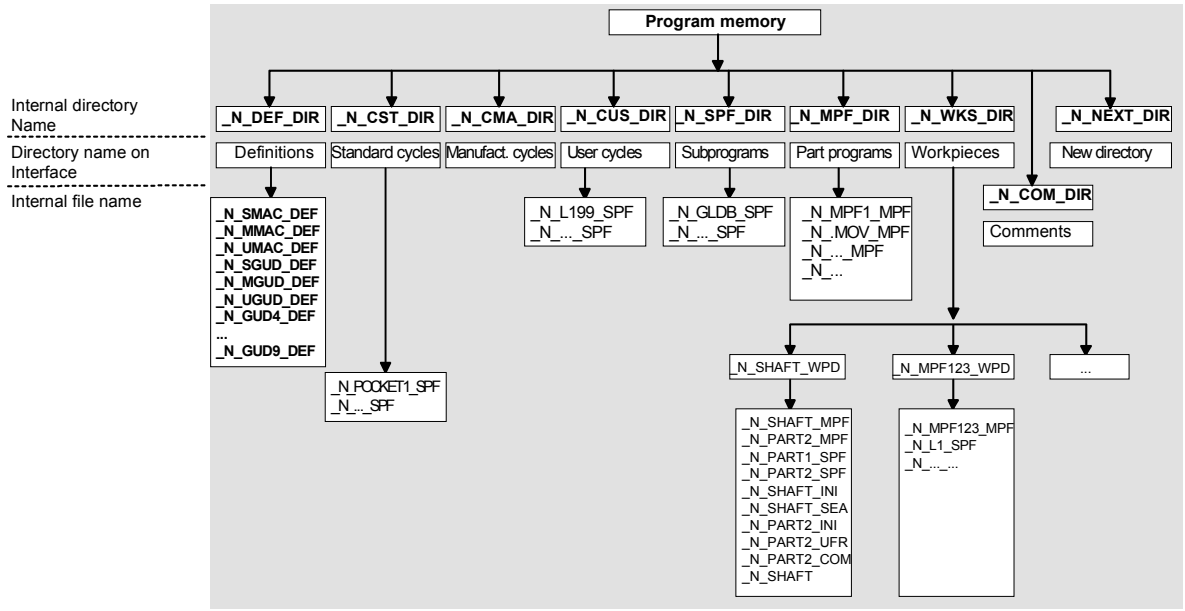
6.6.1 Overview

Program management

To allow you to handle files and programs flexibly, you can organize, store and display them according to different criteria.
The data/programs are stored in the NCK memory by default.
The programs and files can be managed in both the “Program” and “Services” operating areas.

Name of directories	Operating area
• Subprograms	Programs and Services
• Part programs	Programs and Services
• Workpieces	Programs and Services
• Comments	Services
• Definitions	Services
• Standard cycles	Programs and Services
• Manufacturer cycles	Programs and Services
• User cycles	Programs and Services

The following diagram shows an example of directory contents:



© Siemens AG 2008 All rights reserved.

6.6.2 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	Subprogram
name.TEA	Machine data
name.SEA	Setting data
name.TOA	Tool offsets
name.UFR	Zero offsets/frames
name.INI	Initialization file
name.COM	Comment
name.DEF	Definition of global user data and Macros

It is only possible to access the \COM.DIR directories and \DEF.DIR definitions in the “Services” operating area

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

Data unit of the NCK: Data modules 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,
Setting, user, 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

© Siemens AG 2008 All rights reserved.

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	Subprogram
PART2.SPF	Subprogram
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

© Siemens AG 2008 All rights reserved.

6.6.3 Templates

Templates are objects that can be used for creating new workpieces, part programs, subprograms, work offsets, etc., and can be edited and customized to suit the task at hand.
Templates can be designed in different languages and named to allow easy language-specific access to them. If you create a template with a wildcard in the name, the current workpiece name will be inserted into the name when the template is activated.
There is no difference between creating templates and creating workpieces, part programs, subprograms, job lists, work offsets, etc. You can use the administration functions and naming conventions to enter objects you want to use as templates in the template directory.

Templates can be created from the Services operating area directly in the required directories.
Existing templates are offered for selection when you create new objects.

TEMPL
If this string is assigned as part of the template name, it is replaced by the workpiece name when the template is activated. The template is not copied if the name exceeds 24 characters after replacement.

_XX
If the template name terminates in _XX (XX stands for the language code, e.g., _GR for German), only the objects are offered from the template directory where the current language matches the language code in the template name. Templates for different languages can therefore be stored together in the template directory.
Language-specific templates are given priority over non-language-specific templates with the same core name component. Therefore, you can create a template which can be used for all languages for which no language-specific variants are available.

Additional information

Language-specific templates are given priority over non-language-specific templates with the same core name component. Therefore, you can create a template which can be used for all languages for which no language-specific variants are available.

© Siemens AG 2008 All rights reserved.

Sequence of operations

If the template already exists, it will be copied and renamed when you select "New". All files in this directory are also copied into the new directory.

New

Press the "New" softkey.
Specify a name, e.g. "AXIS".
The new "Axis" workpiece is thus created with the following files:
ACHSE.MPF
DATEN.INI
TEST.MPF

6.6.4 Calling a 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.

Sequence of operations

Select workpiece/part program:

Work-pieces
- Workpieces

Part program
- Part programs

Sub-routines
- Subprograms

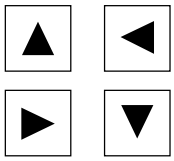
To access the cycle softkeys, press the ETC key.

Standard cycles
- Standard cycles

Manufact. cycles
- Manufacturer cycles

User cycles
- User cycles

© Siemens AG 2008 All rights reserved.



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.

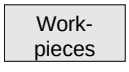
Call a part program:
Use the cursor to select a program in the program overview and press the "INPUT" key.
The text editor is displayed with the file you have selected.

You can now edit the part program.

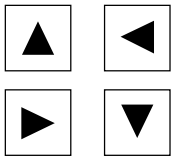
Open workpiece:
The workpiece directory is opened and the programs it contains displayed on the screen.
You can create various types of files such as main programs, tool offsets, etc., in the workpiece directory.

6.6.5 Create workpiece directory

Sequence of operations

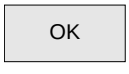


The current overview of all workpiece directories appears on the screen.



Position the cursor on the required directory and open it.
An overview listing the data and programs that have already been created in the directory will be displayed. If no data is available, the program overview will be blank.
For each file, the file name, file type, length, date of creation/last change are displayed.

Press the "New" softkey. A dialog window will appear.
Enter the name of the new directory using the alphanumeric keyboard.
The file type is .WPD (workpiece directory).



Confirm with the "OK" softkey. A new directory is set up in the workpiece overview.

© Siemens AG 2008 All rights reserved.

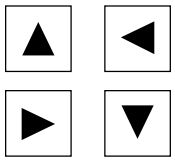
6.6.6 Creating a program/data in a workpiece directory

This section explains how you can create a new file, e.g. for a part program.

Sequence of operations

Work-pieces

The current overview of all workpiece directories appears on the screen.



Position the cursor on the required directory and open it.
An overview listing the data and programs that have already been created in the directory will be displayed. If no data is available, the program overview will be blank.
For each file, the file name, file type, length, date of creation/last change are displayed.

New

Press the "New" softkey to display a dialog window in which you can enter the new file name.
The workpiece/program names may be a maximum of 24 characters in length. You can use any letters (except umlauts), digits or the underscore symbol (_).
Any templates for the specified data type in the Templates directory will be offered for selection.
The matching file type is automatically assigned in this case.

The following file types can be entered:

File type	Description
.MPF	Main Program File
.SPF	Subprogram (subprogram file)
.TOA	Tool offset (Tool Offset Active)
.INI	Initializing data
.COM	Comment file
.GUD	User data (global)
.TEA	NC machine data (testing data active)
.SEA	Setting data active (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

© Siemens AG 2008 All rights reserved.

OK or INPUT

Then press the "OK" softkey or the "Input" key.

Create part programs in the part program/subprogram directory:

Part program or Sub-routines

You can set up main programs and subprograms by opening directories "Part programs" and "Subprograms".

New

Press the "New" softkey to display a dialog window in which you can enter the new program name.
The matching file type is automatically assigned in this case.

6.6.7 Selecting a program for execution

Function

Workpieces and part programs must be selected for execution before you press the Cycle Start key.

Sequence of operations

Select a program:

Part program

Use the cursor keys to select a program in the

▼

program overview, e.g., part programs, and

Program selection

then press the "Program Selection" softkey.
The program name is displayed in the "Program name" window at the top right.

Select a workpiece:

A workpiece can be selected for machining in a channel.

Work-pieces

Use the cursor keys to select the workpiece

▼

in the workpiece overview and then

Workpiece selection

press the "Selection" softkey.

© Siemens AG 2008 All rights reserved.

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 NCK). 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 the files
SHAFT.SPF and SHAFT.MPF.

Selection

When you select workpiece directory SHAFT.WPD you implicitly select program SHAFT.

Find path for program call

If the call path is not explicitly specified in the part program when a subprogram (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.type	Workpiece/standard directory MPF.DIR
2.	/SPF.DIR / name.type	Global subprograms
3.	/CUS.DIR / name.type	User cycles
4.	/CMA.DIR / name.type	Manufacturer cycles
5.	/CST.DIR / name.type	Standard cycles

© Siemens AG 2008 All rights reserved.

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`	Subprograms
5.	`/CUS.DIR` / name.`SPF`	User cycles
6.	`/CMA.DIR` / name.`SPF`	Manufacturer cycles
7.	`/CST.DIR` / name.`SPF`	Standard cycles

Once you have selected the workpiece, change back to the Machine, Automatic operating area and press the "Cycle start" key 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!

6.6.8 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.

Sequence of operations

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

Press the "Change enable" softkey.
A cross indicating "Enable issued" appears behind the workpiece or part program.

© Siemens AG 2008 All rights reserved.

(X) Enable issued (program can be executed)

( ) No enable issued (program must not be executed)

Additional 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.

6.6.9 Copy file and paste

Function

This subsection explains how files can be copied.

Sequence of operations

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 "Insert" softkey, enter a new name and confirm with "OK".

The file types are automatically adjusted in the global part program and the global subroutine directory.

Additional information

- Only files can be stored in a workpiece directory but 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.

© Siemens AG 2008 All rights reserved.

6.6.10 Renaming a file

Function

As regards files, you can alter their name as well as the associated file type.

Sequence of operations

Position the cursor on the file you want to rename.

Rename

The "Rename" dialog window opens.

Enter the new name.

When you rename a workpiece, you can also change the file type with the Toggle key.
File types are matched automatically in the part program and subprogram directories.

There are two ways of renaming files:

- Renaming the workpiece directory
- Renaming a directory in the workpiece directory

Rename 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 as B.XXX, i.e. the extension is retained.

Rename a directory 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.

© Siemens AG 2008 All rights reserved.

6.6.11 Deleting a file

Function

This section explains how you can delete workpieces or files.

Sequence of operations

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.

Delete

The prompt "Do you really want to delete the file?" appears.

OK

Confirm your input.

Additional 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.

© Siemens AG 2008 All rights reserved.

6.7 Memory info

Function

You can call a display showing the total available NCK memory space.

Sequence of operations

Memory info

Press the “Memory info" softkey to display a window section containing the current utilization:

NC memory		KB
Directories	Number of directories/files, of which • total possible • already assigned • still vacant	Size of the NC files and directories
Files		

6.8 EXTCALL

EXTCALL can be used to reload a program from the HMI in "Execution from external source" mode. This enables all programs that can be accessed via the directory structure to be reloaded and executed.

Limitations

The following supplementary conditions must be taken into account with EXTCALL calls:

- With EXTCALL, you can only call files with the MPF or SPF extension.
- The files and paths must adhere to the following NCK conventions: Max. 25 characters for the name, 3 characters for the file extension.
- Set option for network drive and set option for memory expansion on the CompactFlash Card (since otherwise the card cannot be accessed).
- Only permissible for single-channel machines

Note

In a part program, there must be at least one other NC block between two sequential EXTCALL calls, with a processing duration of at least one second.

© Siemens AG 2008 All rights reserved.

Important

Direct execution of a USB FlashDrive is not recommended, as no protection is provided against failure or accidental removal of the USB FlashDrive. Disconnecting it during tool machining will lead to machining stopping immediately and, thus, to the workpiece being damaged.

Search sequence

If the EXTCALL command is used to call a geometry subroutine in a technology program, the control searches for this subroutine in the possible storage locations in a certain order. If the called program is not found in a storage location, the next storage location is searched in accordance with the search order.

The following sequence is followed and completed during a search

1. Search in the storage location specified via setting data item SD SD42700 $SN_SC_EXT_PROG_PATH, if a path is specified.
2. Search in the HMI user memory of the CompactFlash Card. Prior to searching, a test is performed to check whether a targeted EXCALL call with an absolute path specification is involved.

EXTCALL with use of setting date

The channel-specific setting date enables you to specify a storage location for the call via EXTCALL. If this is the case, the program called with the EXTCALL command is searched for directly in the storage location specified in the command. A file of the same name must exist in the specified storage location for a program to be found. If such a file does not exist, the search continues at the next storage location.

Example:

SD42700 $SN_SC_EXT_PROG_PATH =
„/user/sinumerik/data/prog/kanal1“
...
EXTCALL "geoprog.spf“
A search is performed for the file “/user/sinumerik/data/prog/kanal1 geoprog.spf.

The setting data can be used to perform a targeted search for the program.

© Siemens AG 2008 All rights reserved.

EXTCALL using the HMI user memory

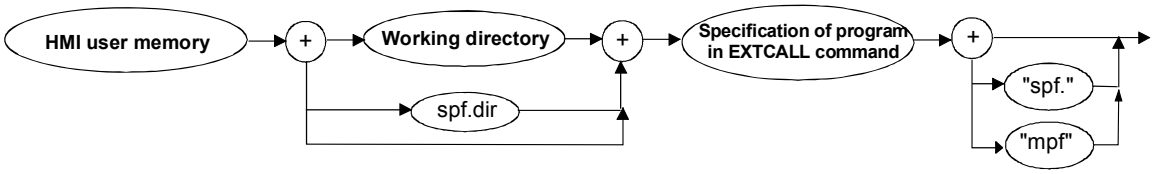
The following requirements must be fulfilled:

1. The technology program is selected in the NCK main memory for execution.
2. The channel-specific setting date has not been set with a directory path or the search in the storage location specified with the setting date was not successful.
3. The HMI user memory option is set. As a result, a “Local drive” softkey is available in the "Program" and "Services" areas and can be used to display and manage the content of the HMI user memory.

The HMI user memory is subdivided into

- Part programs (mpf.dir),
- Subprograms (spf.dir) and

Workpieces (wks.dir) with the respective workpiece directories (.wpd)



HMI user memory:
The path on the CF card to the HMI user memory (/user/sinumerik/data/prog)

Working directory:
The directory in which the technology program is stored (MPF:DIR, WKS.DIR / xxx.WPD). If the program is not found in the working directory, the search is continued in the spf.dir. Last, the root directory of the HMI user memory is searched.

EXTCALL commands:
EXTCALL “geoprog“
EXTCALL “geoprog.spf“
EXTCALL “wks.dir /xxx.wpd/geoprog.spf“
EXTCALL “spf.dir/geoprog.spf”
EXTCALL “mydir.dir/geoprog.spf”

If you do not indicate a program extension, the first attempt is with the extension .spf and then with .mpf

Place the geometry program in the HMI user memory as a subprogram in the /spf.dir directory. Use unique program names, i.e., assign different names for different geometry programs in the control. Call the program from the technology program with the following commands:

```
EXTCALL (“geoprog“),  or
EXTCALL (“geoprog.spf“), or
EXTCALL (“_N_GEOPROG_SPF“).
```

The “geoprog.spf” program from the "/spf.dir" directory is executed in the HMI user memory.

© Siemens AG 2008 All rights reserved.

EXTCALL with targeted call

In addition to use of the search mechanism described above, you can also call a specific subroutine with the EXTCALL command by indicating the storage location of the subroutine in the EXTCALL command.

Examples:

- HMI user memory:

```
EXTCALL
("/user/sinumerik/data/prog/spf.dir/myextsub.spf")
```

The “myextsub.spf” program stored in the "/spf.dir" directory in the HMI user memory is called.

```
EXTCALL
("/user/sinumerik/data/prog/wpd.dir/mywpd.dir/myextsu
b.spf")
```

The “myextsub.spf” program stored in the “mywpd.dir” workpiece in the "/wks.dir" workpiece directory in the HMI user memory is called.

```
EXTCALL
("/user/sinumerik/data/prog/mydir.dir/myextsub.spf")
```

The “myextsub.spf” program stored in the "/mydir.dir" directory you created in the HMI user memory (option!) is called.

- Network drive:

```
EXTCALL ("//MyServer/MyDir/myextsub.spf")
```

The "myextsub.spf" program stored in the "MyDir" directory release by the "MyServer" server.

Syntax: //Computer name/Release name/Program path

- USB drive to TCU:

```
EXTCALL ("//TCU://TCU_1:/X203/MyDir/myextsub.spf")
```

The “myextsub.spf” program stored in the "/MyDir" directory on the USB memory device that is connected to the TCU named TCU_1 at interface X203 is called.
Syntax: //TCU name/Name of USB interface/program path
USB interface: X203 or 204

■

© Siemens AG 2008 All rights reserved.

Services operating area

© Siemens AG 2008 All rights reserved.

7.1 Function

Function

The "Services" Operating Area provides the following functions:

- Read programs and data in/out
- Manage data
- Series machine startup
- Error log

7.2 Directory structure

All programs and files are saved on the CompactFlash card in a fixed directory structure.

7.2.1 Directories

The following directory contains special files:

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 NCK 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 NCK 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 and executed in the working memory of the NCK

© Siemens AG 2008 All rights reserved.

7.2.2 Data selection

The following selection of file types can be read in or out from the CompactFlash card, or a configured network connection:

File type	Description
.MPF	Part program (main program file)
.SPF	Subprogram (subprogram 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	Setting data active (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	Folder
.DEF	Definitions data
.CEC	Sag/angularity
.QEC	Quadrant error compensation
.EEC	Measuring system error compensation

© Siemens AG 2008 All rights reserved.

Keywords/directories in the hierarchical file structure

The different files are made available for transmission under the following headings:

- NCK active data
  - all
  - machine data
    - all
    - NC-specific
    - channel-specific
    - axis-specific
  - setting data
    - all
    - NC-specific
    - channel-specific
    - axis-specific
  - option data
    - all
    - NC-specific
    - channel-specific
    - axis-specific
  - global user data
    - all
    - NC-specific
    - channel-specific
  - tool and magazine data
    - all
    - tool data
    - magazine data
  - protection zones
    - all
    - NC-specific
    - channel-specific
  - R variables
    - all
    - channel 1/2
  - zero point offset
    - all
    - channel 1/2
  - sag/ angular compensation
  - quadrant error compensation
    - all
    - axis 1 to axis 28
  - SSFK/ measuring system error compensation
    - all
    - axis 1 to axis 28

© Siemens AG 2008 All rights reserved.

- Display machine data
- Workpieces
- Part programs
- Subroutines
- User cycles
- Manufacturer cycles
- Standard cycles
- Cycle storage
  - Standard cycles
  - Cutting cycles
  - Iso cycles
  - Measuring cycles
  - Manufacturer cycles
  - User cycles
- Comments
- Definitions
  OEM
- System data
  - ASUP1
  - ASUP2
  - IBN
  - OSTORE1
  - OSTORE2
  - Version
- Protocol
  - Log book
  - Communication error log
  - Machine configuration (in preparation)

If your control includes additional directories, these can be found in the file tree.

© Siemens AG 2008 All rights reserved.

7.3 Formats for saving and importing data

Path name

The path name is automatically entered when files are saved (archived). Files are stored on the CompactFlash Card and can also be backed up there in various paths (user, oem).
The entire path in which the file was stored is specified 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.

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 copied or backed up as archives in the following formats:

- IBN archive PC/binary format
- All others: punched tape/ASCII format

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`.

Reading in 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.

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.

© Siemens AG 2008 All rights reserved.

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.
1. Files in punched tape format can be edited with the text editor.
2. Files can be set up externally in punched tape format provided that they are formatted in compliance with the format specified below.
3. 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 Set             NL          ;contents of file 1
2nd block           NL
...                 NL
last block          NL

%2nd file name
;$PATH=2nd path name        ;can be included
1st Set             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.

© Siemens AG 2008 All rights reserved.

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 NCK 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 = part program SHAFT123 **or** %flange3_MPF = part 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=`".

© Siemens AG 2008 All rights reserved.

<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	Part programs
%SPFn	_N_SPFn_SPF	/_N_SPF_DIR	Subprograms
%Ln	_N_SPFn_MPF	/_N_SPF_DIR	Subprograms
%*	_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`

© Siemens AG 2008 All rights reserved.

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

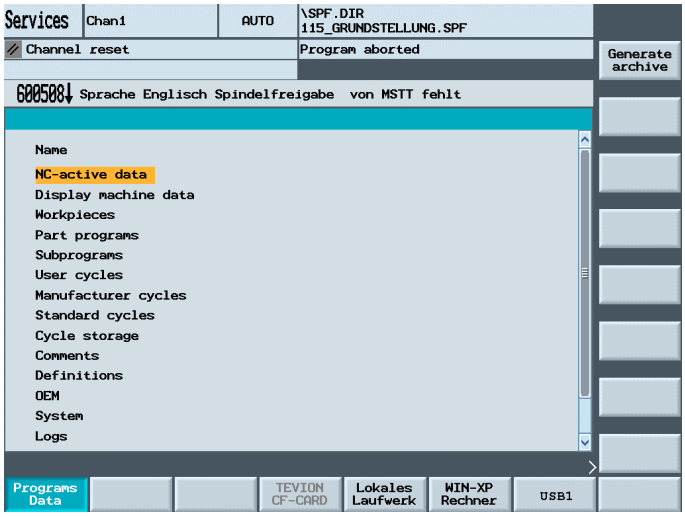
Files which contain characters that cannot be displayed/binary format can only be saved in PC format.

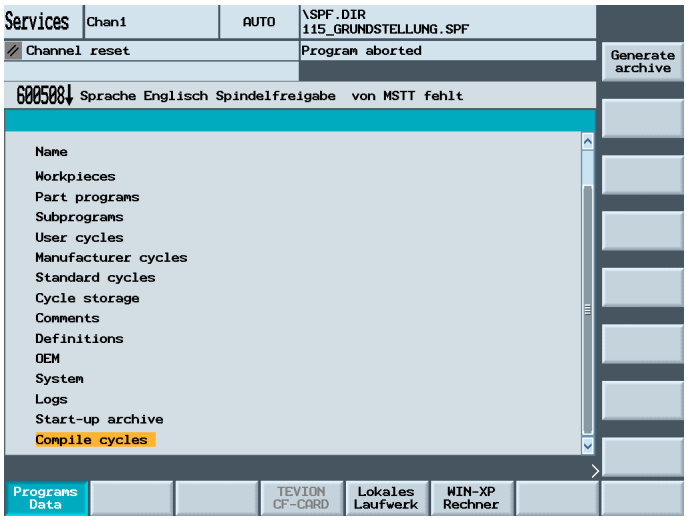
- If you save files that have been backed up in PC format and then edit them with a text editor, you will not be able to import them again. The file cannot be edited because the checksum will then no longer be correct.
- Start-up and update data must always be saved in PC format.

© Siemens AG 2008 All rights reserved.

7.4 Services main screen

All transferable data/programs are displayed in the "Services" main screen.





Horizontal softkeys

Programs Data

All programs and files are displayed.

TEVISON CF-CARD **Local drive** **WIN-XP computer** **USB1**

Configured "logical drives",
e.g. TEVISON CF CARD
Local drive (CompactFlash card)
WIN-XP computer (network connection)
USB1(USB connections to the TCU)
Another four drives (if configured) can be accessed with the ETC key

Log

You obtain information about the data transmission which has taken place.

© Siemens AG 2008 All rights reserved.

For set option:

Compile cycles

Loadable compiler cycles (with the .elf extension) can be loaded with the "Compile cycles" softkey.
The loaded files can be displayed in the “Diagnostics” operating area.

Vertical softkeys

Create archive

The selected programs or files are prepared for upload or download by pressing the “Create archive” softkey.

7.4.1 Series machine startup

Function

With series start-up, you can archive the current software on your machine and for example import it to other machines later.
You can select all NCK data, or just the drive or 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.

Notes

Loadable compile cycles cannot be backed up or read in using the series machine startup archive.

Sequence of operations

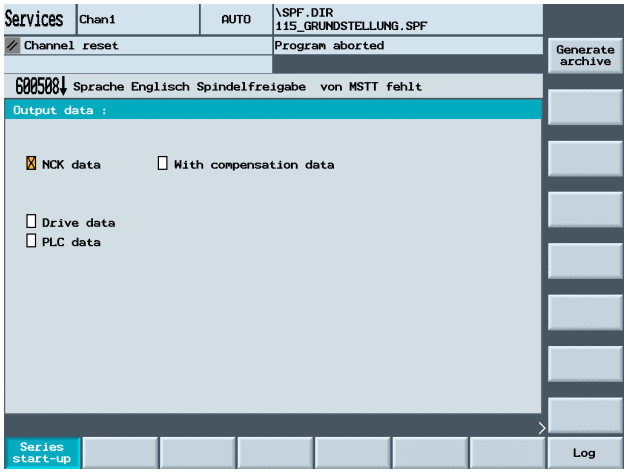
>

Press the ETC key in the basic display.

Series IBN

Press the “Series IBN” softkey and select the data that you want to archive.

© Siemens AG 2008 All rights reserved.



Create archive

Press the "Create archive" softkey.

OK

All configured drives will be offered to you in a window section for you to make your selection. Choose the drive, e.g. local drive, and press "OK".

OK

The system automatically sets up storage directory "card/user/sinumerik/data/archive" and displays a list of the folders it contains for you to choose. Use the arrow keys to choose a folder, e.g. "NCK_Aktuell", the press "OK" again.

OK

You will then see the storage directory and the system will suggest a default name for the archive, e.g. NC.ARC.
You can still change this name.
Press the "OK" softkey to start the archiving operation.

Log

You can display information about the data transfer operation by pressing softkey "Log".

© Siemens AG 2008 All rights reserved.

7.4.2 Output error/transmission log

Function

A log listing data that have been imported and exported can be output in the Services operating area.

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"	File not complete in archive format
"User Abort"	Transmission terminated with "Stop" key. Archive: Not completely transmitted, last file not stored. Punched tape: Completeness cannot be checked, last file stored.
"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"	Files are not in archive format. Please copy with Copy/Insert.
"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.

Sequence of operations

Log

You can display information about the data transfer operation by pressing softkey "Log".

© Siemens AG 2008 All rights reserved.

7.4.3 Read ISO programs in/out

Function

ISO programs can be read in and out in punched tape format.
The output is only possible in punched card/ASCII format.

Additional 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=<Pfad> 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

Create archive

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 "Create archive". 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 on the NCK as main programs. You must set the read-in directory every time you import programs to the “Services" area by selecting → "Import archive" → Select path → "OK". If "Path from workpiece/archive" is selected, ISO programs are stored in the selected directory (e.g. workpiece xxx) or in the default NCK directory (MPF.DIR); DIN programs are stored in accordance with their specified path.

© Siemens AG 2008 All rights reserved.

ISO punched tape with two ISO programs:

```
%
O1026(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(2. SPEED RANGE)
N20 G00 G80 G90 G40 G17
(...)
N200 G53 Y0. Z0.
N210 M30
```


© Siemens AG 2008 All rights reserved.

Readout

Create archive

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 "Create archive". 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
;$PATH=/_N_WKS_DIR/_N_TEST_WPD
N40 G01 X150 Y150 Z150 F6000
N50 G90 G0 X0 Y0 Z0 G53
; ...
N500 G02 z100 x50 k-50 i0
N510 z50 x100 k0 i50
M30     ;transition from Siemens prog. to Siemens prog.
%_N_TEST2_MPF
;$PATH=/_N_WKS_DIR/_N_TEST_WPD
N40 G01 X150 Y150 Z150 F6000
```

© Siemens AG 2008 All rights reserved.

```
; ...
M30     ;transition from Siemens prog. to ISO prog.
%
O1127(ANGLE)
N10(2. 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
;$PATH=/_N_WKS_DIR/_N_TEST_WPD
N40 G01 X150 Y150 Z150 F6000
N50 G90 G0 X0 Y0 Z0 G53
; ...
N500 G02 z100 x50 k-50 i0
N510 z50 x100 k0 i50
M30     ;transition from Siemens prog. to Siemens prog.
%_N_TEST2_MPF
;$PATH=/_N_WKS_DIR/_N_TEST_WPD
N40 G01 X150 Y150 Z150 F6000
; ...
M30     ;transition from Siemens prog. to ISO prog.
%_N_1127_MPF
;$PATH=/_N_WKS_DIR/_N_TEST_WPD
(ANGLE)
N10(2. SPEED RANGE)
N20 G00 G80 G90 G40 G17
(...)
N200 G53 Y0. Z0.
N210 M30
```

The difference is irrelevant for archives in binary format.

Additional 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.

■

© Siemens AG 2008 All rights reserved.

Diagnostics operating area

© Siemens AG 2008 All rights reserved.

8.1 Diagnostics main screen

A display headed "Alarms" appears when you select the operating area "Diagnosis".

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.
Clearing 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 the "Service displays" softkey.
PLC	Information on the current status of the PLC memory locations.
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.

Sequence of operations

Alarms:

Alarms	The alarm overview displays all active alarms with alarm numbers, date, clearance criteria and text.

© Siemens AG 2008 All rights reserved.

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

Press "Reset" key.

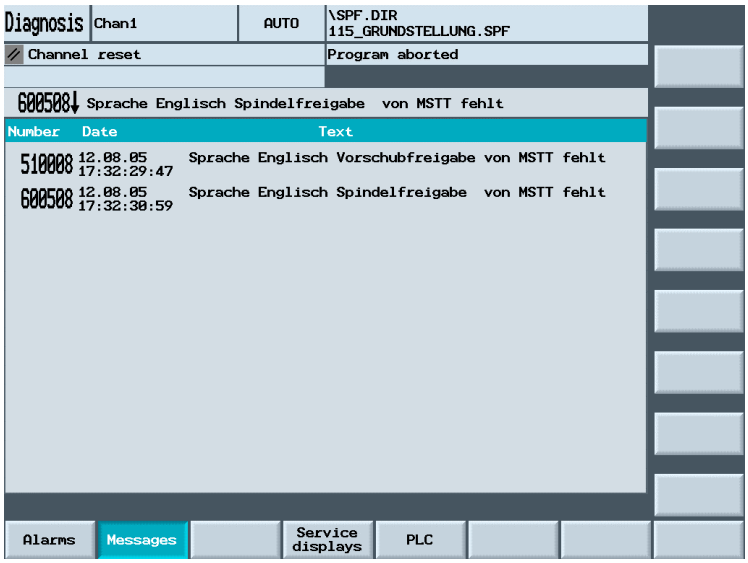
Press "Alarm Cancel" key.

Alarm is cleared with the "Cycle Start" key.

Alarm is cleared with the "Recall" key (message box).

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.

Messages:



The PLC operating messages that are not (correctly) acknowledged must be displayed.

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:

or

© Siemens AG 2008 All rights reserved.

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 ½, 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 Function Manual, Basic Functions; Diagnostics Tools (D1)

Sequence of operations

Service displays

Select the menu "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 PAGE UP or

You can page up and down with the "Page" keys or

▲ ▼

with the direction keys.

Axis + Axis -

The service values of the next (+) and the previous (-) axis are displayed.

© Siemens AG 2008 All rights reserved.

8.3.2 Service drive

Function

The information contained in the "Drive service" 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
- displaying 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

Function Manual, Basic Functions; Diagnostics Tools (D1)

Sequence of operations

Service displays

Select the menu "Service displays".
The horizontal softkey bars change.

Service drive

The "Drive service" window displays information about the axis drive together with axis name and number.

PAGE DOWN PAGE UP or ▲ ▼

You can page up and down with the "Page"

keys or the direction keys.

Drive + Drive -

The service values of the next (+) and the previous (-) drive are displayed.

© Siemens AG 2008 All rights reserved.

8.3.3 Safety Integrated service

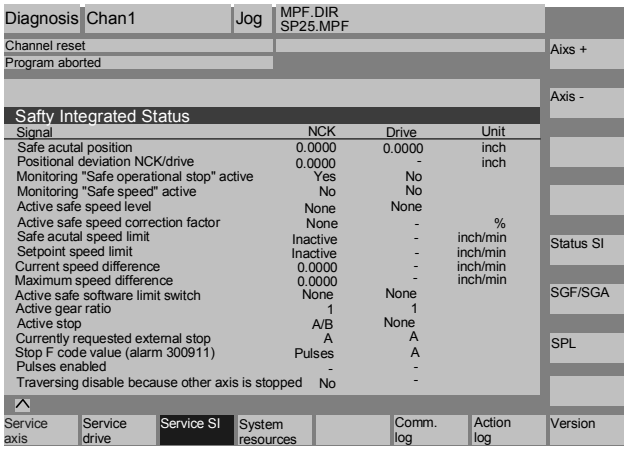
Function

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 Function Manual 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

© Siemens AG 2008 All rights reserved.

Sequence of operations

Service displays

Select the menu "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.

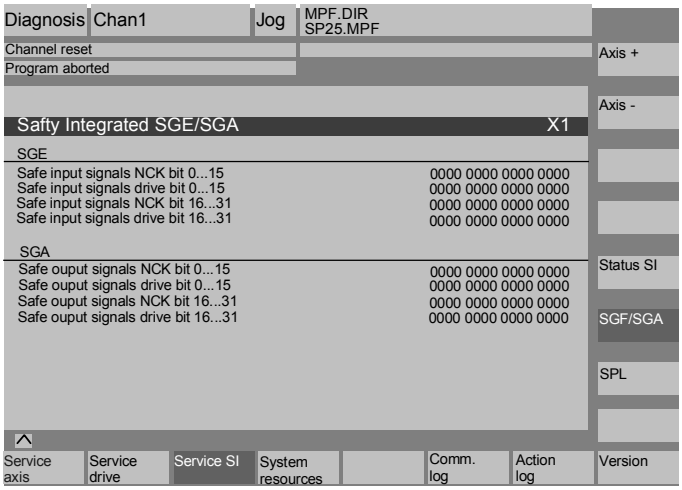
SGE/SGA

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.

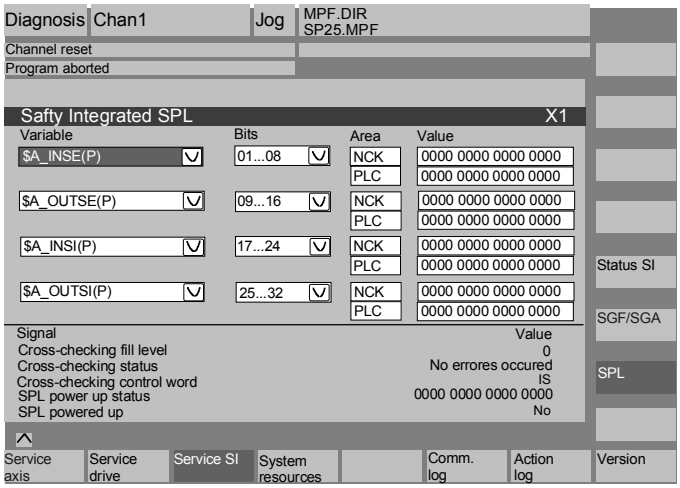
SGE/SGA



The available signals can be seen in the above screen.
The Status SI vertical softkey takes you to the Status SI screen, SPL opens the Safe Programmable Logic screen.

© Siemens AG 2008 All rights reserved.

SPL



In the "Variable" selection box, you can select:
$A_INSE(P) corresponds to simultaneous selection of
$A_INSE top row; NCK origin and
$A_INSEP bottom row; PLC origin
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

KDV level (CDC Crosswise data comparison)
Cross-checking status
Cross-checking control word
SPL power up status
SPL powered up

© Siemens AG 2008 All rights reserved.

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
Bit 9: NCK cross-checking has been started
Bit10: PLC cross-checking has been started
Bit11: Cyclic SPL checksum checking active
Bit12: All SPL protective mechanisms active

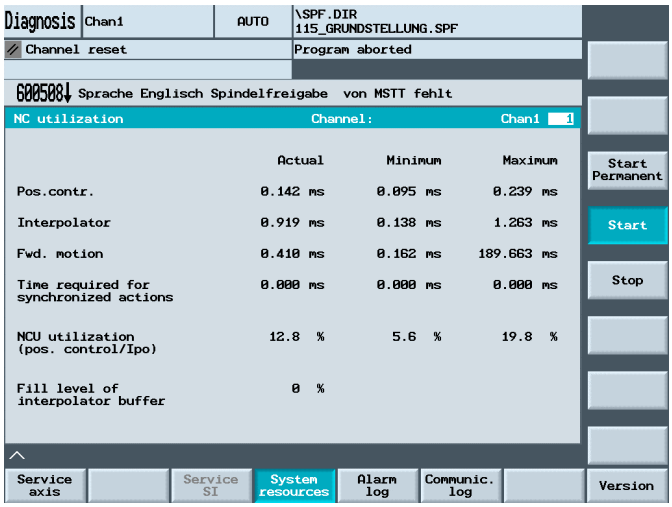
8.3.4 Displaying system resources

Function

You can display the system resources (utilization display) that are currently being used for the individual NCU areas:

Net and gross runtimes of

- position controller,
- interpolator and
- preprocessing.



© Siemens AG 2008 All rights reserved.

Sequence of operations

Service displays

System resources

Press the "Service displays" and the "System resources" softkeys.
The "NC utilization" screen is displayed.
The following minimum/maximum total data for the servo, IPO cycle and preprocessing are displayed:

- Net runtime in ms
- Gross runtime in ms
- Level of the IPO buffer in percent
- Total capacity utilization in percent

Start Permanent

The current values are constantly updated.

Stop

Start

The display update can be halted with the "Stop" softkey, the displayed values are updated again with the "Start" softkey.

8.3.5 Alarm log

Function

A log containing the alarms that have occurred so far is displayed.

Alarm log

The alarm overview displays all active alarms together with their date, time, alarm number, and description.

The error log file is principally used by the control manufacturer (Siemens) as a diagnostic tool.

Save as...

The log file can be read out with the “Save as..." softkey.

© Siemens AG 2008 All rights reserved.

8.3.6 Communication error 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." softkey.

The error log file is principally used by the control manufacturer (Siemens) as a diagnostic tool for communication errors.

Save as...

The communication error log file can be read out with the “Save as..." softkey.

8.3.7 Traverse log

Function

The log for individual operating sequences is shown.

Machine manufacturer

Please note information supplied by machine tool manufacturer!

Sequence of operations

Traverse log

The log contains all the data that you set in the "IBN" operating area, together with the date, time, number and description.

Save as...

The log file can be saved on the CompactFlash card via the “Save as..." softkey.

References

A detailed description of the traverse log and the setting of the data to be logged are contained in the following documentation:
Commissioning Manual, Commissioning HMI-Embedded;
Commissioning HMI-Embedded (IM2): Traverse Log".

© Siemens AG 2008 All rights reserved.

8.4 Calling the version display

Function

The version data of the installed system software are output in the Version display.
In the event of a fault, the version data can be read out via the configured (logical) drives and sent to the Hotline support staff.

Sequence of operations

Version

When you open the "Version" window, you can access several different areas. You can use the horizontal softkeys to view the version data.

NCU Version

Version data for the NCU (system, version stamp, date, module code) is shown.

Details

With the softkey, you receive additional information about the installed software.

HMI Version

Version data for the HMI Embedded (name, version, date, time, length).

Cycle versions

Press the “Cycle versions” softkey to change the vertical softkey bar. You can select name, type, length and version of the following cycle areas with additional softkeys:

- Package overview
- All cycles
- User cycles
- Manufacturer cycles
- Standard cycles

Definitions

Version display or definitions (name, type, length, version)

Compile cycles

Only if option selected, version display for compile cycles.
See Section: “Display compile cycles”

Hardware version

Version display for the installed hardware.

PAGE DOWN PAGE UP

Use the "Page" keys to scroll up and down.

Save versions

Data are saved and can be read out.

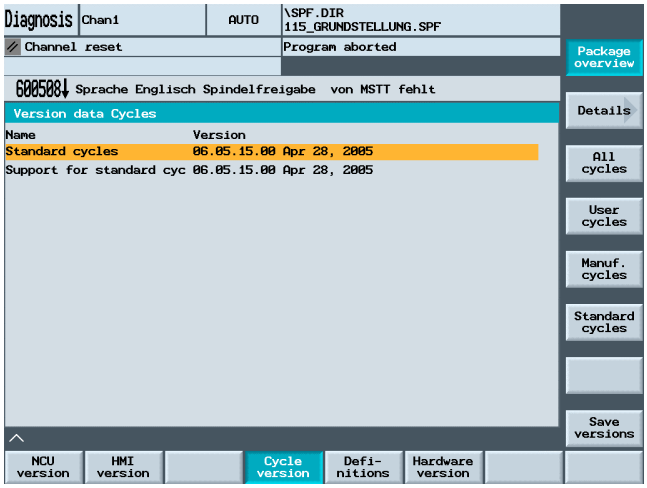
© Siemens AG 2008 All rights reserved.

8.4.1 Displaying the version display for the cycles

Function

The following version screens can be displayed and used to diagnose the cycle states:

- Overview of the cycle packages available on the control.
- Details of the individual packages.
- An overview of all available cycles.
- Separate overview for user, manufacturer and standard cycles.



Sequence of operations

Cycle versions

Package overview

The cycle packages available on the NCK can be displayed in a separate "Cycle version data" overview by pressing the "Version data cycles" horizontal softkey. The "Package overview" vertical softkey is included automatically.

Related cycles can be grouped together in packages and listed in a package list. The complete package is assigned a package name and a version identifier. The file type of the cycle package list is .cyp (cycle package). Package lists are named cyc_xxx.cyp and the following standard names are defined:

© Siemens AG 2008 All rights reserved.

Predefined package lists	Package list	Cycle package
	cyc_sc.cyp	Standard cycles
	cyc_scs.cyp	Cycle support (for standard cycles)
	cyc_mc.cyp	Measuring cycles
	cyc_mcs.cyp	Measuring cycle support
	cyc_mj.cyp	Measuring in JOG
	cyc_sm.cyp	ShopMill
	cyc_st.cyp	ShopTurn
	cyc_mt.cyp	ManualTurn
	cyc_c950.cyp	Extended stock removal
	cyc_c73.cyp	Pocket surfaces with islands
	cyc_iso.cyp	ISO compatibility for cycles
	cyc_cma.cyp	Manufacturer cycles (predefined name)
	cyc_cus.cyp	User cycles (predefined name)

Details

Select a package in the package overview and press the "Details" vertical softkey. The following details for the selected package appear in the "Version data" overview:

- Package name Name
- Package type Type
- Load status Loaded
- Package length Length
- Storage directory Directory (in data management)
- Date
- Version entry Version

All cycles

All available cycles of the types .com and .spf from the user cycle, manufacturer cycle, and standard cycle directories are displayed in the "Version data" overview irrespective of the packages.

User cycles

List of user cycles (CUS.DIR),

Manufact. cycles

manufacturer cycles (CMA.DIR),

Standard cycles

standard cycles (CST.DIR).

© Siemens AG 2008 All rights reserved.

8.4.2 Display version screen of loadable Compile cycles

Function

If compile cycles are available in the NCK, you can display them in a separate version screen. Besides the current version (name, extension, type, date, time, length) and the start address (path/CF card), the access authorization for "reading, writing, enabling, listing and deleting" are displayed.

Sequence of operations

Compile cycles are loaded when you initiate an NCU reset and can be displayed.

Compile cycles

Press the "Compile cycles" softkey.

All loaded files of the type **.elf** are displayed in the "Version data compile cycles" overview.

Details

If you press the "Details" softkey, you will receive more information.

Path	:\card\....\Loadable compile cycles			
Name	:CCMCSC	Date:	Time	
extension	:ELF	Length:	Loaded: ☐	
Type	:Loadable compile cycle			
Information about access rights:				
Read:	Write:	Enable:	List:	Delete:
Additional version information:				
Version: MCSC Coupling axes Machine			Time	Date
Advance version of compile cycle (Preliminary)				
Interface: 001.001@Interfaces=002.000 @TChain=001.000				

© Siemens AG 2008 All rights reserved.

8.4.3 Displaying the version display for the definitions

Function

You can display the definition files available on the NCK in a separate overview.

Sequence of operations

Definitions

If you press the "Definitions" softkey, you will receive all the definition files from the data management DEF.DIR directory.

8.4.4 Displaying the hardware version

Function

All the version data for all the hardware components on the machine is shown in one overview.

Sequence of operations

Hardware version

If you press the "Hardware Version" softkey, you receive the version information for the individual components of the machine, e.g. Mainboard, FPGA, BIOS, SINUMERIK PLC, CF Card

© Siemens AG 2008 All rights reserved.

8.5 PLC

8.5.1 General information

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)
Outlets:	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 doublewords) C = character (ASCII character)

Operand	Example	descriptive elements	Writing	Format	Value	Area
Inputs		Yes	Yes			0-127
	I 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	
Bit memory		Yes	Yes			0-255
	M 60.7			B	1	
	MB60			B	1101 0110	
	MW 60			H	B8	
				D	180	
				C		
Timers	T20	Yes	No			0-31
				B		
				H		
				D		

© Siemens AG 2008 All rights reserved.

Operand	Example	descriptive elements	Writing	Format	Value	Area
Counter instructions	C20	Yes	Yes	B H D		0-31
Data block **Data byte**	DB3.DBB9	Yes	Yes	H D B C	A 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 configuration of the parameters can endanger human life and cause damage to the machine.

8.5.2 Change/delete operand value

Function

The values of operands can be changed.

Sequence of operations

PLC | PLC status

Press the "PLC" and "PLC status" softkeys.
The first operand screen form appears.
The vertical softkey bar changes.

Operand + | Operand -

You can increase or decrease the address of the operand by 1 place at a time.

© Siemens AG 2008 All rights reserved.

Input aid

Input help:
Instead of entering the operands and formats directly in the fields, you can also generate them by pressing the "Input aid" softkey and then the softkeys which appear subsequently.
The following softkeys are available:

"DB"	-> .DBX, DBB, DBW and DBD
"M Marker"	-> B Byte, W Word, D Double-word
"I Input"	-> B Byte, W Word, D Double-word
"Q Output"	-> B Byte, W Word, D Double-word
"T Time"	-> T
"C Counter"	-> C

When the cursor is in the "Format" input field, the following vertical softkeys are available:
B Binary, D Decimal, H Hexadec., F Float, C Character.

Delete

The entries for the selected operand (formats and values) are deleted when you confirm the prompt "Do you really want to delete all entries" with the "OK" softkey.
Press the "Recall" key if you do not want to delete them.

Change

Cyclic updating of the values is interrupted.

Abort

Cyclic updating is continued; the entered values are not transferred to the PLC.

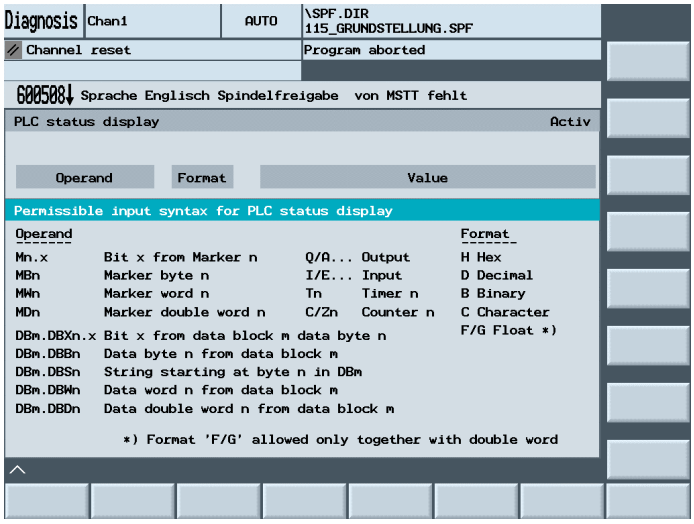
Accept

The entered values are transferred to the PLC. Cyclic updating is continued.

Additional information

Press the "information key".
A description of the permissible input syntax for the PLC status display is overlaid.

© Siemens AG 2008 All rights reserved.



8.5.3 Select/create operand form 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.

© Siemens AG 2008 All rights reserved.

Sequence of operations

The function can also be selected in the “Services” operating area.

PLC

Press the "PLC" softkey.
The first operand screen form appears.
The vertical softkey bar changes.
Press softkey "Read in operands" (via application diskette).

without calculation

OK

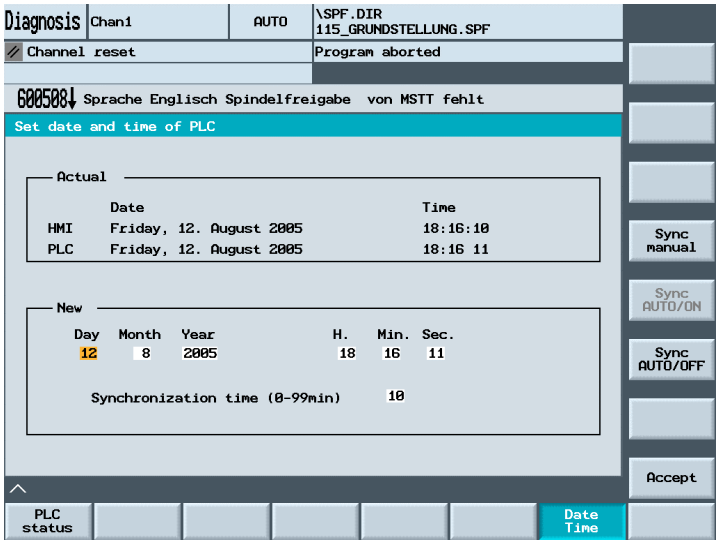
Position the cursor on the operand form you wish to find and confirm by pressing softkey "OK".
The screen form you selected is imported into the PLC status display.

8.5.4 Set 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.

Sequence of operations



PLC

Press the "PLC status” softkey.
The horizontal and vertical softkey bars change.

© Siemens AG 2008 All rights reserved.

Date/ Time

When you press the "Date/Time" softkey, you 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.
The synchronization can be verified in the "Current time:" output field.

References

Commissioning Manual, Commissioning Base software and HMI-Embedded; Commissioning HMI-Embedded (IM2): Set date/time

The set values are retained when the control is rebooted.

Additional 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

Please follow the instructions of the machine manufacturer.

© Siemens AG 2008 All rights reserved.

8.6 Activating remote diagnostics

The remote diagnostics function is an option.
If this option is set, it is possible to control and influence the operation of an NCU from a remote PC, as well as to transmit data.

Remote diagnosis

When you select the "Remote diagnostics" softkey, the vertical "Start" and "End" softkeys appear. In this way you start or end the remote diagnostics.

Instructions on configuring and operating remote diagnostics are contained in the following documentation:

References Commissioning Manual, Commissioning Base software and HMI-Embedded; Commissioning HMI-Embedded (IM2): Remote diagnostics, RCS Host-Embedded/RCS View-Embedded

Machine manufacturer Please follow the instructions of the machine manufacturer.

■

© Siemens AG 2008 All rights reserved.

© Siemens AG 2008 All rights reserved.

Commissioning operating area

© Siemens AG 2008 All rights reserved.

9.1 Commissioning main screen

Danger

Changes in the Commissioning operating area have a significant influence on the machine. Incorrect configuration of the parameters can endanger human life and cause damage to the machine.

Access to certain menus in the "Commissioning" operating area can be protected by keylock switch or password.

This Chapter describes functions which the machine operator can perform on the basis of his or her access rights.

For more detailed information on the subject of startup for

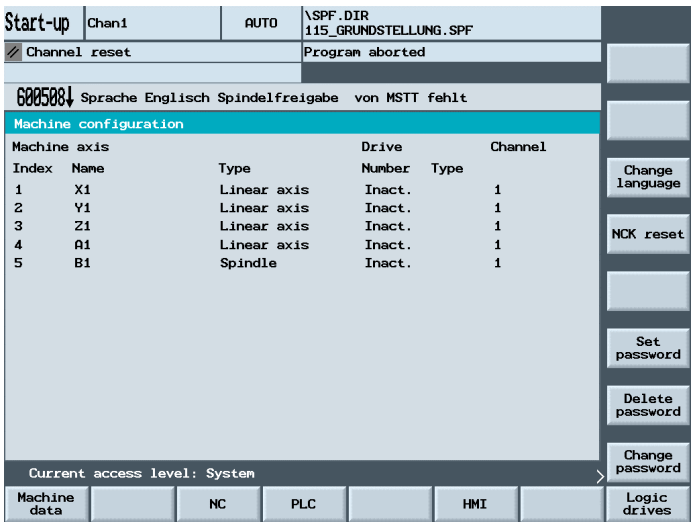
- System personnel
- Machine manufacturer
- Service personnel
- Machine users (machine setters)

Please refer to the following documentation:

References

Commissioning Manual, HMI-Embedded; Commissioning HMI-Embedded (IM2):
TCU Base software (IM7)
Equipment Manual NCU
Operator Component Manual
CNC Commissioning Manual: NCK, PLC, Drive

The "Machine configuration" window is displayed in the "Start-up" basic display.



© Siemens AG 2008 All rights reserved.

Horizontal softkeys

Machine data
Enables you to edit the machine data for all areas.

NC
Here you can look at the NC settings for NC power up and the NC address, and change them if necessary.

HMI
Here you can enter the basic settings for the operator panel front (e.g. color settings).

Logical drives
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, such as a CompactFlash Card, USB FlashDrive, etc.

Vertical softkeys

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 an NCK power ON/Reset for the NCK and drive.

Set password
You can set a password.

Delete password
You can delete a password.

Change password
You can change your password.

© Siemens AG 2008 All rights reserved.

9.2 Displaying machine data

Access to the machine data operating area can be controlled by keylock switch or password.

Function

Areas

The machine data is 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 is provided for each of these areas in which you can view and edit machine data.

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 data
- Unit of the machine data
- Activation

If the machine data does 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 (meter/second squared): Acceleration
rev/s**3	rev/s^3 (revolution/second to the power of 3): Change in rate of acceleration for rotating axis
kg/m**2	kgm^2 (kilogram/meters squared): Moment of inertia:
mH	mH (millihenry): Inductance
Nm	Nm (Newton meters): Torque
us	µs (microseconds): Time

© Siemens AG 2008 All rights reserved.

µA	µA (microamperes): Electric current
µVs	µVs (microvolt-seconds): Magnetic flux
userdef	User-defined: The units are defined by the user.

The right-hand column indicates when a machine data becomes effective:

- so = immediately effective
- cf = when confirmed via the "Activate MD" softkey
- re = reset
- po = POWER ON (NCK Power On reset)

Sequence of operations

Machine data

Pressing the "machine data" soft-key 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-spec. 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 configuration of the parameters can endanger human life and cause damage to the machine.

Drive MD

Drive-specific machine data ($MD_)

Display MD

Operator panel front machine data ($MM_)

VSA

Machine data for the feed drive,

© Siemens AG 2008 All rights reserved.

HSA

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.

Find next

After pressing the “Find next” softkey, you move from one datum to the next in the machine data list.

9.2.1 Display options: Display filters

Function

The purpose of display 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 word filter (previously a reserve)
3. Each area has a maximum of 13 groups (group 14 is reserved for Expert mode (see below), bit 15 is reserved for add-ons).

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:

© Siemens AG 2008 All rights reserved.

Criteria	Check
1. Access authorizations	If the level of access authorization is not sufficient, the MD is not displayed. Otherwise criterion 2 is checked.
2. Display 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 display 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 display filter, then criterion 6 is checked. If none of the group bits is set and "All others" is not selected in the display 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.

Sequence of operations

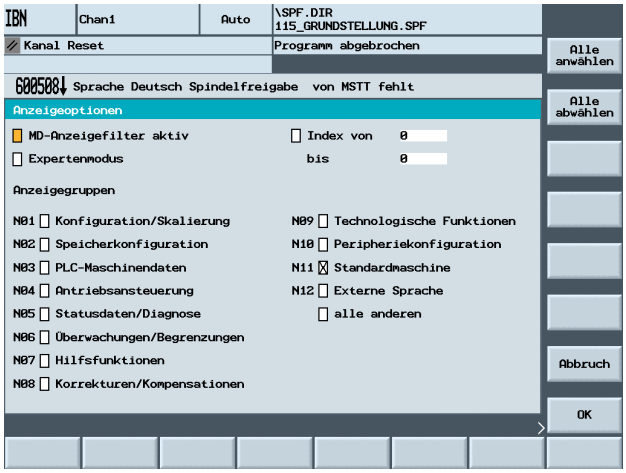
Machine data

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.

© Siemens AG 2008 All rights reserved.



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 without saving.

9.3 NC settings

Function

NC

Here you can look at the NC settings for NC power up, view and change the NC address if necessary, and power up again.
Setting a key position for the startup switch with appropriate access rights.

IBN switch

After pressing the "IBN switch" softkey, you can choose from the following via the vertical softkeys:

Normal booting

- Normal booting.

The following safety query will appear:
"Do you want to perform an NCK reset?"
Confirm with “Yes” or abort with “No”.

IBN mode

- Start-up with default values (IBN mode):

The following safety query will appear:

"An NCK overall reset will delete all the data in the buffer memory (SRAM) and load the default machine data.
Do you really want to perform an NCK overall reset?"

© Siemens AG 2008 All rights reserved.

If you answer "Yes", the overall reset will start. If you answer no, the operation will be aborted.

NCK-Shut-Down

When you press this softkey, the NCK will begin a controlled shutdown without a safety query.

9.4 PLC

Function

You can get information about the current statuses of the following PLC memory locations, and change them if necessary:

- Set date/time
- Import operands

This function is also available in the “Diagnosis” operating area.

Changes can only be made to the PLC operands with the appropriate password.

The procedure for setting the date/time and reading in PLC operands is described in
Chapter 8, “Diagnostics operating area”, in the section titled “PLC”.

References

Commissioning Manual, Base software and HMI-Embedded;
Commissioning HMI-Embedded (IM2): Setting the date/time

Danger

Changes in the states of PLC memory locations have a major impact on the machine. Incorrect configuration of the parameters can endanger human life and cause damage to the machine.

© Siemens AG 2008 All rights reserved.

9.5 Changing the HMI user interface

Function

You have the option of making the following settings for the HMI Embedded user interface.

- Color settings on screen
- Specify language selection
- Activate traverse logging
- Open Editor

Sequence of operations

HMI

Pressing the "HMI" softkey changes the horizontal and vertical softkey bars.
The following **submenus** are available via the horizontal softkeys:

9.5.1 Color setting

Function

You can customize your color settings for the HMI-Embedded user interface.

Sequence of operations

Colors

If you press the "Colors" softkey, the "Color settings" menu appears: "Standard" is preset in the "Color schema" window section.

Use the Select key to select the "Users" field.
You will now be presented with the individual HMI user interfaces (e.g. background, window border, title text, CursorEditor etc.). When you have selected an area, you will receive a color palette from which you can select the colors you want using the direction keys and the "INPUT" key.
You can change the various areas one after the other. When you have made your settings for all the user interfaces,

© Siemens AG 2008 All rights reserved.

Save

press "Save".
To accept the settings, HMI-Embedded shuts down and restarts.

OK

Return to the menu by pressing "OK".

Load default

Press the "Load default" softkey to return to the preset color scheme.

Last version

Press the "Last Version" softkey to restore the last color settings that were saved.

References

Commissioning Manual, HMI-Embedded; Commissioning HMI-Embedded (IM2):

9.5.2 Language selection

Function

Language selection

In general, you can use two languages at the same time during operation.
German, English, French, Italian, Spanish and Chinese are available as standard.

Change Language

To toggle between the two selected languages, press the "Change Language" softkey in the basic display.

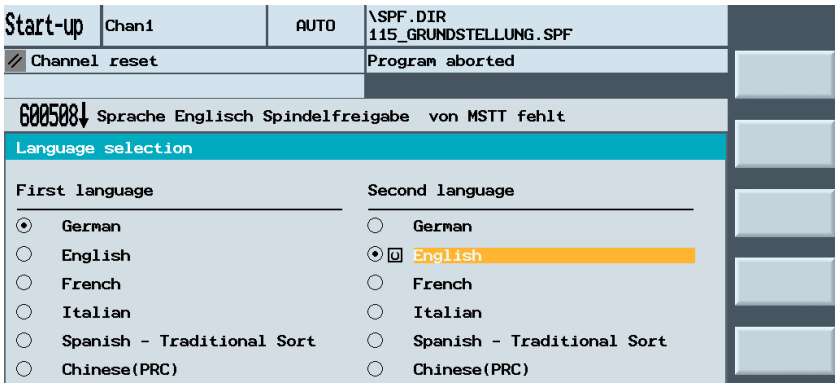
To select the languages you want, proceed as follows:

Sequence of operations

Language default

Languages

If you press the "Languages" softkey you will receive a list in which German, English, Spanish, French and Italian can each be selected as the 1st or 2nd language.



© Siemens AG 2008 All rights reserved.

User the cursor key to select the two languages you want in the first and second columns of the list.

Confirm your choice with the “Select” selection key.

OK

Confirm with "OK".

References

Commissioning Manual, HMI-Embedded; Commissioning HMI-Embedded (IM2): Installing/selecting language

9.5.3 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:

If logging is switched on, you can display the following data:

- Interrupts
- Keys
- Channel status/override
- Windows IDs

OK

You can save the settings via the "OK" softkeys.

The log is output in the "Diagnosis" operating area.

References

Commissioning Manual, Base software and HMI-Embedded; Commissioning HMI-Embedded (IM2): Traverse log

© Siemens AG 2008 All rights reserved.

9.5.4 Editor (HMI only)

Editor (HMI only)

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

The description of this function and the settings for the editor are included in chapter 2: “General operating sequences”, in the section titled “Editor”, and in the following document:

References

Commissioning Manual, Base software and HMI-Embedded; Commissioning HMI-Embedded (IM2): Functions, setting editor

© Siemens AG 2008 All rights reserved.

9.6 Setting up logical drives

Function

On HMI-Embedded, the links and connections for CompactFlash Card, network and USB connections can be used with logical drive definitions.

Logical drive

You can set up up to eight connections. With these defined softkeys, you can display directories and programs from various storage locations.

These softkeys are available in the "Program", "Services" and “Commissioning” operating areas, depending on the configuration of your system.

The procedure used to set up the additional softkeys is described in:

References

Commissioning Manual, Base software and HMI-Embedded; Commissioning HMI-Embedded (IM2): Setting up network connections

9.7 Initiate NCK reset

Function

NCK Reset

After pressing the “NCK Reset” softkey, the safety query "Do you want to initiate an NCK reset?“ appears.
Press the “Yes“ softkey to restart the NCK and drives.
If you press the "No" softkey, you will return to the previous screen.

9.8 Set, delete, change password

Function

The control has a protection level system for enabling data areas.
There are access levels 0 to 7; where

- 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 keylock switch settings.

© Siemens AG 2008 All rights reserved.

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 by default.

Protection level	Locked by	Area
0	Password	Siemens
1	Password	Machine manufacturer
2	Password	Commissioning engineer, service
3	Password	end user
4	Key-operated switch setting 3	Programmer, machine setter
5	Key-operated switch setting 2	Qualified operator
6	Key-operated switch setting 1	Trained operator
7	Key-operated switch setting 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 keylock switch position is ignored.

Sequence of operations

Set password

You can set a password via the vertical softkeys,

Delete password

delete a password and

Change password

to change the selection.

Please refer to the following documentation for a more detailed description:

References

Commissioning Manual, Base software and HMI-Embedded;
Commissioning HMI-Embedded (IM2): Protection levels

© Siemens AG 2008 All rights reserved.

9.9 Display or modify system resources

Function

You can display the system resources currently in use for the NC and HMI Embedded areas (utilization display).

Sequence of operations

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.

Additional 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

Commissioning Manual, Base software and HMI-Embedded; Commissioning HMI-Embedded (IM2): Displaying and modifying system resources

© Siemens AG 2008 All rights reserved.

9.10 Licensing

A suitable license is needed in order to use the options.
With the right level of access authorization, you can use the user interface to set up any licenses you have acquired, as well as display an overview of the options and their licenses.
Please refer to the following documentation for a more detailed description of this subject.

References

Commissioning Manual, Base software and HMI-Embedded;
Commissioning HMI-Embedded (IM2): Licensing
CNC Commissioning Manual: NCK, PLC, Drive

■

© Siemens AG 2008 All rights reserved.

© Siemens AG 2008 All rights reserved.

Maintenance

© Siemens AG 2008 All rights reserved.

10.1 Operating data

Operating data

	Value
Air humidity, humidity class to DIN 40040	F
Atmospheric pressure	860 to 1080 hPa
Protection against physical contact protection class to DIN VDE 0160	I
Degree of protection according to DIN 40050	
• Front of operator panel	IP 54
• Back of operator panel	IP 00
• Front of machine control panel	IP 54
• Back of machine control panel	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 washing-up 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:

- Diluted mineral acids
- Bases
- Organic hydrocarbons
- Detergent solutions

Plastic material used

The plastic material used on the front of the SINUMERIK 840D is suitable for applications on machine tools.

It is resistant to:

1. Greases, oils, mineral oils
2. Bases and lyes
3. Detergent solutions and
4. Alcohol

Solvents such as chlorinate hydrocarbons, benzene, esters and ethers should be avoided!

■

© Siemens AG 2008 All rights reserved.

Appendix

© Siemens AG 2008 All rights reserved.

A Abbreviations

µC	Micro Controller
A	Output
AS	Automation system
ASCII	American Standard Code for Information Interchange Amerikanische Code-Norm für den Informationsaustausch
ASIC	Application Specific Integrated Circuit: User switching circuit
ASUB	Asynchronous subroutine
AuxF	Auxiliary Function
AV	Job planning
BA	Mode
BB	Ready
BCD	Binary Coded Decimals: Decimal numbers encoded In binary code
BCS	Basic Coordinate System
BIN	BINary files
BIOS	Basic Input Output System
BOT	Boot files: Boot files for SIMODRIVE 611D
BP	Basic program
C Bus	Communication bus
CAD	Computer-Aided Design
CAM	Computer-Aided Manufacturing
CNC	Computerized Numerical Control Computerized Numerical Control

© Siemens AG 2008 All rights reserved.

COM	Communication
COR	Coordinate rotation
CP	Communications Processor
CPU	Central Processing Unit: Central processing unit
CR	Carriage Return
CRC	Cutter radius compensation
CRT	Cathode Ray Tube picture tube
CSB	Central Service Board: PLC module
CSF	Function plan (PLC programming method)
CTS	Clear To Send: Signal from serial data interfaces
CUTOM	CUTter radius cOMpensation: Tool radius compensation
DAU	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: Movement of the rotary axis via the shortest path to the absolute position within one revolution
DCD	Data Carrier Detect
DDE	Dynamic Data Exchange
DIN	Deutsche Industrie Norm (German Industry Standard)
DIO	Data Input/Output: Data transfer display
DIR	DIRectory: Folder
DLL	Dynamic Link Library

© Siemens AG 2008 All rights reserved.

DOE	Data transmission equipment
DOS	Disk Operating System
DPM	Dual-Port Memory
DPR	Dual-Port RAM
DRAM	Dynamic Random Access Memory
DRF	Differential Resolver Function: Differential revolver function (handwheel)
DRY	DRY run: Dry run feedrate
DSB	Decoding Single Block: Decoding single block
DTE	Data Terminal Equipment
DW	Data word
E	Input
EIA code	Special punched tape code, number of holes per character always odd
ENC	Encoder: Actual value encoder
EPROM	Erasable Programmable Read Only Memory
Error	Error from printer
FB	Function block
FBS	Slimline screen
FC	Function Call: Function block in the PLC
FDB	Product database
FDD	Floppy Disk Drive
FDD	Feed Drive
FEPROM	Flash-EPROM: Read and write memory

© Siemens AG 2008 All rights reserved.

FIFO	First In First Out: Memory that works without address specification and whose data are read in the same order in which they were stored.
FIPO	Fine InterPOlator
FM	Function Module
FPU	Floating Point Unit Floating Point Unit
FRA	Frame block
FRAME	Data block (FRAME)
FST	Feed Stop: Feed stop
GUD	Global User Data : Global user data
Hardware	Hardware
HD	Hard Disk Hard disk
HEX	HEXadecimal number
HHU	Handheld unit
HMI	Human Machine Interface
HMS	High-resolution Measuring System
I/O	Input/Output
I/R	Infeed/Regenerative Feedback unit (power supply) on the SIMODRIVE 611(D)
IBN	Commissioning
ICA	Interpolatory Compensation Interpolatory compensation
IF	Drive module pulse enable
IK (GD)	Implicit communication (global data)
IM	Interface Module Interconnection module
IMR	Interface Module Receive: Interconnection module for receiving data

© Siemens AG 2008 All rights reserved.

IMS	Interface Module Send: Interconnection module for sending data
INC	Increment : Increment
INI	Initializing Data : Initializing data
IPO	Interpolator
IS	Interface signal
ISA	Industry Standard Architecture
ISO	International Standard Organization
ISO code	Special punched tape code, number of holes per character always even
JOG	JOGging: Setup mode
K1 .. K4	Channel 1 to channel 4
K_v	Servo gain factor
LAD	Ladder diagram (PLC programming method)
LCD	Liquid Crystal Display: Liquid crystal display
LEC	Leadscrew error compensation
LEDs	Light Emitting Diode : Diode
LF	Line Feed – end of a block, produced by pressing the ENTER key.
LR	Lageregler
LUD	Local User Data
MB	Megabyte
MC	Measuring Circuit
MCP	Machine control panel
MCS	Machine Coordinate System
MD	Machine data

© Siemens AG 2008 All rights reserved.

MDA	Manual Data Automatic: Manual input
MLFB	Machine-readable product designation
MMC	Man-Machine Communication: User interface on numerical control systems for operator control, programming and simulation
Mode group	Mode group
MPF	Main Program File: NC part program (main program)
MPI	Multi-Point Interface Multiport Interface
MS	Microsoft (software manufacturer)
MSD	Main Spindle Drive
NC	Numerical Control: Numerische Steuerung
NCK	Numerical Control Kernel: NC kernel with block preparation, traversing range, etc.
NCU	Numerical Control Unit: NCK hardware unit
NRK	Name for the operating system of the NCK
NURBS	Non-Uniform Rational B-Spline
OB	Organization block in the PLC
OEM	Original Equipment Manufacturer
OP	Operator Panel
OP	Operator Panel: Operating equipment
OPI	Operator Panel Interface
OPI	Operator Panel Interface: Interface for connection to the operator panel
OPT	Options: Options
OSI	Open Systems Interconnection: Standard for computer communications

© Siemens AG 2008 All rights reserved.

P bus	Peripheral Bus
PC	Personal Computer
PCIN	Name of the SW for data exchange with the control
PCMCIA	Personal Computer Memory Card International Association plug-in memory normalization board
PG	Programming device
PLC	Programmable Logic Control: Programmable logic control
PLC	Speicherprogrammierbare Steuerung
PMS	Position measuring system
POS	Positioning
RAM	Random Access Memory: Program memory which can be read and written into
REF	REFerence point approach function
REPOS	REPOSition function
RISC	Reduced Instruction Set Computer: Type of processor with small instruction set and ability to process instructions at high speed
ROV	Rapid Override : Input correction
RPA	R-Parameter Active: Memory area in NCK for R-NCK for R parameter numbers
RPY	Roll Pitch Yaw: Rotation type of a coordinate system
RS-232-C	Serial interface (definition of the exchange lines between DTE and DCE) of a computer
RTS	Request To Send: RTS, control signal of serial data interfaces
SBL	Single Block : Single Block
SCK	Software Configuration Kit (tool supplied with screen-kit)
SD	Setting Data

© Siemens AG 2008 All rights reserved.

SDB	System Data Block
SFB	System Function Block
SFC	System Function Call
SK	Softkey
SKP	SKiP: Skip block
SLS	Setting Data Active: Identifier (file type) for setting data
SM	Stepper Motor
SPF	Sub Program File : Subprogram
SR	Subprogram
SRAM	Static RAM (non-volatile)
SRT	Transformation ratio
SSI	Serial Synchronous Interface: Synchronous serial interface
STL	Statement list
SW	Software
SYF	SYstem Files 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 Tool offset
TO	Tool offset
TOA	Tool Offset Active: Identifier (file type) for tool offsets

© Siemens AG 2008 All rights reserved.

TRANSMIT	TRANSform Milling Into Turning: Coordinate conversion on turning machine for milling operations
TRC	Tool Radius Compensation
UFR	User Frame: Zero point shift
UI	User interface
WCS	Workpiece coordinate system
Window ID	System-wide unique number of a window or a screen
WOP	Workshop-oriented Programming
WPD	Workpiece Directory: Workpiece directory
ZO	Zero point shift
ZOA	Zero Offset Active: Identifier (file type) for work offset data

© Siemens AG 2008 All rights reserved.

B Terms

Important terms are listed in alphabetical order. The "->" symbol precedes terms, which are explained in a separate entry in this list.

A

A spline

The Akima spline runs under a continuous tangent through the programmed interpolation points (3rd order polynomial).

Absolute dimension

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 by a 7-level system of access restrictions:

- Three password levels for system manufacturers, machine manufacturers and users and
- Four keylock switch settings, 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.

Alarms

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. Alarms and messages are displayed separately.

1. Alarms and messages in the part program
   Alarms and messages can be displayed directly from the part program in plain text.
2. Alarms and messages from PLC
   Alarms and messages relating to the machine can be displayed directly from the PLC program in plain text. No additional function block packages are required for this purpose.

© Siemens AG 2008 All rights reserved.

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.

Archive
Reading out data and/or directories to an **external** memory device.

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

© Siemens AG 2008 All rights reserved.

B

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).

Backlash compensation Compensation of a mechanical machine backlash, e.g., backlash on leadscrews. Backlash compensation can be entered separately for each axis.

Backup Saving the memory contents to an external memory device.

Back-up 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.

Backup 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 data bytes.

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 the part is machined.

Block "Block" is the term given to any files required for creating and processing programs.

Block A section of a -> parts program terminated with a line feed. A distinction is made between -> main blocks and -> subblocks.

Block search The block search function allows any point in the part program to be selected, at which machining must start or be continued. The function is provided for the purpose of testing part programs or continuing machining after a program abort.

Booting Loading the system program after power on.

© Siemens AG 2008 All rights reserved.

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.

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.

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.

Chassis 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.

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 language 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.

© Siemens AG 2008 All rights reserved.

Compensation value Difference between the axis position measured by the position sensor and the desired, programmed axis position.

Connecting 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.

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 definable tolerance band as a measure of contour accuracy. Overloading of the drive, for example, may result in an unacceptably large following error. In such cases, an alarm is output and the axes are stopped.

Coordinate system See -> machine coordinate system, -> workpiece coordinate system.

Cutter radius compensation Contour programming assumes that the tool is pointed. Since 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 repeated machining operation on the -> workpiece
An NC code generated via "Expand operator interface" or a geometry process consists of several lines.

Cycle support The available cycles are listed in the "Cycle support" menu in the "Program" operating area. Once the desired machining cycle has been selected, the parameters required for assigning values are displayed in plain text.

© Siemens AG 2008 All rights reserved.

D

Data block

1. Data unit of the -> PLC, which the -> HIGHSTEP programs can access.
2. Data unit of the -> NC: Data modules contain data definitions for global user data. These data can be initialized directly when they are defined.

Data transfer program PCIN

PCIN is an auxiliary program for transmitting and receiving CNC user data, e.g., parts programs, tool offsets, etc., via the serial interface. The PCIN program can run under MS-DOS on standard industrial PCs.

Data word

A data unit, two bytes in size, within a -> data block.

Diagnostics

1. Operating area of the control.
2. The control has both a self-diagnostics program as well as test functions for servicing purposes: status, alarm and service displays.

Digital input/output module

Digital modules are signal transducers for binary process signals.

Dimensions specification, metric and inches

Position and pitch values can be programmed in inches in the machining program. The control is set to a basic system regardless of the programmable dimensional specification (G70/G71).

DRF

Differential Resolver Function: NC function which generates an incremental work offset in Automatic mode in conjunction with an electronic handwheel.

Drift compensation

When the CNC axes are in the constant motion phase, automatic drift compensation is implemented in 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 excellent machining precision even at high tool path feedrates. The feedforward control can be individually selected and deselected for each axis in the parts program.

© Siemens AG 2008 All rights reserved.

E

Editor The editor makes it possible to create, edit, extend, join, and import programs/texts/program blocks.

Electronic handwheel Electronic handwheels can be used to traverse the selected axes simultaneously in manual mode. The handwheel clicks are analyzed by the increment analyzer.

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 precise destination point. The -> part program continues execution at the next block.

External zero offset Zero offset specified by the -> PLC.

F

Fast retraction from contour When an interrupt is received, it is possible to initiate a motion via the CNC machining program, which allows the tool to be lifted quickly 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. (SINUMERIK 840D).

Feed override The current feedrate setting entered via the control panel or by the PLC is overlaid on the programmed feedrate (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, such as 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.

© Siemens AG 2008 All rights reserved.

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

General reset The following -> CPU memories are erased by a general reset operation:
- the -> working memory
- the read/write area of the -> load memory
- the -> system memory
- the -> backup memory

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/subprogram Each global main program/subprogram may appear only once under its name in the directory. It is not possible to use the same program name in different directories with different contents as a global program.

H

Helical interpolation The helical interpolation function is ideal for machining internal and external threads using form milling cutters and for milling lubrication grooves. The helix comprises two movements:
1. Circular movement in one plane
2. 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). High-speed, program-driven switching functions can be initiated via the digital CNC outputs (SINUMERIK 840D).

HIGHSTEP Summary of the programming options for the -> PLC in the AS300/AS400 system.

© Siemens AG 2008 All rights reserved.

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 measuring system

Measuring 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

Travel path length specification based on number of increments. The number of increments can be stored as a -> setting data or selected with 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 file

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.

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.

© Siemens AG 2008 All rights reserved.

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.

Inverse time feedrate

On SINUMERIK 840D controls, it is possible to program the time required to traverse the path of a block instead of the feedrate speed 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 traversed in JOG mode by means of the direction keys. Other functions available in JOG mode are -> reference point approach, -> repositioning and -> preset (setting an actual value).

K

Key switch

S7-300: The keyswitch is the mode selector switch on the -> CPU. The keylock switch is operated by a removable key.

Keywords

Words with a specific notation, which have a defined meaning in the programming language for -> parts programs.

K_v

Servo gain factor, a control variable in a control loop.

© Siemens AG 2008 All rights reserved.

L

Languages
The user guidance display texts and the system messages are available in five system languages (diskette):
German, English, French, Italian, and **Spanish**.
The user can select **two** of the listed languages at a time 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
With CPU 314 of the -> PLC, the load memory is the equivalent of 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.

© Siemens AG 2008 All rights reserved.

M

Machine Operating area of the control.

Machine axes Physically existent axes 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 directly influence the machine tool via the PLC.

Machine coordinate system A coordinate system, which is related to 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 technique Grouping of a set of instructions under a single identifier. The identifier in the program refers to the grouped set 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.

MDA Control operating mode: Manual Data Automatic. In the MDA mode, individual program blocks or block sequences with no reference to a main program or subprogram can be input and executed immediately afterwards through actuation of the NC start key.

Measuring circuits 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, but including no more than 5 spindles.

© Siemens AG 2008 All rights reserved.

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. Alarms and messages are displayed separately.

Metric measurement system

Standardized system of units: for lengths in millimeters (mm), meters (m), etc.

Mirroring

Mirroring reverses the signs of the coordinate values of a contour, with respect to an axis. It is possible to mirror with respect to more than one axis at a time.

Mounting rail

A mounting rail is used to attach the modules of an S7-300.

Mounting system

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.

Multipoint interface

The multipoint interface (MPI) is a 9-pin sub-D port. 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 "Multipoint Interface MPI" parameter block of the CPU contains the -> parameters, which define the properties of the multipoint interface.

© Siemens AG 2008 All rights reserved.

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. These supplements have the same meaning as the words with respect to block format. Identifiers must be unique. It is not permissible to use the same identifier for different objects.

NC Numerical Control: It incorporates all the components of the machine tool control system: -> NCK, -> PLC, -> COM.
Note: CNC (computerized numerical control) would be more appropriate for the SINUMERIK 840D: MARS and Merkur controls.

NCK Numerical Control Kernel: Component of the numerical control system, which executes -> part programs and essentially coordinates the movements on the machine tool.

Network A network is the interconnection of several S7-300 and other terminal devices, such as a programming device, by means of -> connecting cables. A data exchange takes place over the network between the connected devices.

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 using the -> "S7 Configuration" S7-Tool.

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). A standard procedure is thus available (SINUMERIK 840D) as an internal control function for all modes of interpolation.

© Siemens AG 2008 All rights reserved.

O

OEM
The scope for implementing individual solutions (OEM applications) for the SINUMERIK 840D has been provided for machine manufacturers, who wish to create their own operator interface or integrate process-oriented functions in the control.

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.

Operating 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.

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 (e.g., when a tool breaks), a program command can be used to retract the tool in a user-specified orientation by a defined distance.

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.

© Siemens AG 2008 All rights reserved.

P

Parameters

The **S7-300** uses two types of parameter:

1. Parameters of a STEP 7 instruction
   A STEP 7 instruction parameter is the address of the operand to be machined or a constant.
2. Parameter of a -> parameter block
   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.

Part 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 given a name consisting of a maximum of 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 feedrate

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:
The PLC user program, the user data and the basic PLC program are stored together in the PLC user memory. The PLC user memory can be expanded up to 96 KB with memory expansions.

PLC programming

The PLC is programmed with **STEP 7** software. STEP 7 programming software is based on the standard **WINDOWS** operating system and incorporates the functionality of STEP5 programming with innovative

© Siemens AG 2008 All rights reserved.

	expansions and developments.
Polar coordinates	A coordinate system, which defines the position of a point on a plane in terms of its distance from the origin and the angle formed by the radius vector with a defined axis.
Polynomial interpolation	Polynomial interpolation provides a means of generating a very wide range of curves, including **straight-line, parabolic and exponential functions** (SINUMERIK 840D).
Positioning axis	Axis that 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	Control is switched off and then switched on again.
Preset	The control zero point can be redefined in the machine coordinate system by means of the Preset function. Preset does not cause the axes to move; instead, a new position value is entered for the current axis positions.
Program	1. Operating area of the control. 2. Sequence of instructions to the control.
Program block	Program blocks contain the main programs and subroutines of 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).

© Siemens AG 2008 All rights reserved.

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 traverse rate of an axis. It is used, for example, to move the tool from rest to the -> workpiece contour or retract the tool from the contour.

Reference point Point on the machine tool used to reference the measuring system of the -> machine axes.

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. Reapproach contour, triggered by operator
   REPOS allows the tool to be returned to the interrupt position by means of the direction keys.

2. Programmed contour reapproach
   A selection of approach strategies are available in the form of program commands: Approach point of interruption, approach start of block, approach end of block, approach a point on the path between start of block and interruption.

Retentive address areas 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. When the rigid tapping function is used, interpolation of the spindle acting as a rotary axis and the drilling axis ensures that threads are cut exactly to the final drilling depth, e.g., tapped blind hole (requirement: spindles in axis operation).

© Siemens AG 2008 All rights reserved.

Rotary axis
Rotary axes apply a workpiece or tool rotation to a defined angular position.

Rotary axis, turning continuously
Depending on the application, the travel range of a rotary axis can be limited to less than 360 degrees or the axis can be continuously turned in both directions. Continuously turning rotary axes are used, for example, for eccentric machining, 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

S7-300 bus
The S7-300 bus is a serial data bus 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.

S7-Configuration
S7-Configuration is a tool with the aid of which modules can be parameterized. S7 Configuration is used to set a variety of -> parameter blocks of the -> CPU and the I/O modules on the -> programming device. These parameters are transmitted to the CPU.

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 fault, the machining operation is interrupted and the drives stopped. The cause of the malfunction is logged and output as an alarm. At the same time, the PLC is notified that a CNC alarm has been triggered.

Scaling
Component of a -> frame, which causes axis-specific scale modifications.

© Siemens AG 2008 All rights reserved.

Serial RS-232-C interface For the purpose of data input and output, MMC modules PCU 50/70 include two RS-232-C interfaces Machining programs and manufacturer and user data can be loaded and saved via these interfaces.

Services Operating area of the control.

Setting data Data, which communicates the properties of the machine tool to the NC control, as defined by the system software.

Simulator module A simulator module is a module

- on which digital input variables can be simulated via control elements and
- Digital output quantities can be displayed

Soft key A key, whose name appears on an area of the screen. The choice of softkeys displayed is dynamically adapted to the operating situation. The freely assignable function keys (softkeys) are assigned defined functions in the software.

Software limit switch Software limit switches define the limits of the travel range of an axis and prevent the slide contacting the hardware limit switches. 2 pairs of values can be assigned on each axis, which can then be activated separately via the -> PLC.

Spindles The spindle functionality is a two-level construct:

1. Spindles: Speed or position-controlled analog spindle drives digital (SINUMERIK 840D)
2. Auxiliary spindles: speed-controlled spindle drives, "auxiliary spindle" function package e.g. for driven tools.

Spline interpolation Using the spline interpolation function, the control is able to generate a smooth curve from just a small number of specified interpolation points along a setpoint contour.

SRT Transmission ratio

© Siemens AG 2008 All rights reserved.

Standard cycles

Standard cycles are provided for machining operations, which are frequently repeated:

- Cycles for drilling/milling applications
- for turning technology

The available cycles are listed in the "Cycle support" menu in the "Program" operating area. Once the desired machining cycle has been selected, the parameters required for assigning values are displayed in plain text.

Subblock

Block prefixed by "N" containing information for a machining step, such as a position parameter.

Subprogram

A sequence of instructions in a -> parts program, which can be called repeatedly with various defining parameters. The subprogram is called from a main program. Every subprogram can be protected against unauthorized read-out and display. -> Cycles are a form of subprogram.

Synchronization

Instructions in -> parts programs for coordination of sequences in different -> channels at specific machining points.

Synchronized actions

1. Auxiliary function output
   While a workpiece is being machined, technological functions (-> auxiliary functions) can be output 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. High-speed auxiliary function output
   For time-critical switching functions, the acknowledgement times for the -> auxiliary functions can be minimized and unnecessary stops in the machining process can be avoided.

Synchronized axes

Synchronized axes take the same time to traverse their path 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 variables

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.

© Siemens AG 2008 All rights reserved.

T

Teach In

Teach-in is a means of creating or correcting part programs. The individual program blocks can be input via the keyboard and executed immediately. Positions approached via the direction keys or handwheel can also be stored. Additional information such as G functions, feedrates or M functions can be entered 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

A part used on the machine tool for machining. Examples include turning tools, milling cutters, drills, laser beams, etc.

Tool offset

By programming a **T function** (5 decades, integer) in the block, you can select the tool. Every T number can be assigned up to nine cutting edges (D addresses). The number of tools to be managed in the control is set at the configuration stage.

Tool radius compensation

In order 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).

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 travel range for linear axes is ± 9 decades. The absolute value depends on the selected input and position control resolution and the unit of measurement (inch or metric).

© Siemens AG 2008 All rights reserved.

U

User 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.

User memory

All programs and data, such as part programs, subroutines, comments, tool offsets, and zero 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).

© Siemens AG 2008 All rights reserved.

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 made/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 made/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 by the distance from the machine zero.

X

Y

Z

Zero/work offset Specification of a new reference point for a coordinate system through reference to an existing zero point and a -> frame.

1. Settable
   SINUMERIK 840D: A configurable number of settable work offsets are available for each CNC axis. The offsets - which are selected by means of G functions - take effect alternately.
2. External
   In addition to all the offsets, which define the position of the workpiece zero, it is possible to superimpose an external zero offset
   - By means of a handwheel (DRF offset) or
   - PLC
3. Programmable
   It is possible to program work offsets for all path and positioning axes by means of the TRANS statement.

■

© Siemens AG 2008 All rights reserved.

I Index

© Siemens AG 2008 All rights reserved.

© Siemens AG 2008 All rights reserved.

© Siemens AG 2008 All rights reserved.

© Siemens AG 2008 All rights reserved.

© Siemens AG 2008 All rights reserved.

© Siemens AG 2008 All rights reserved.

To
Siemens AG

A&D MC MS1
P.O. Box 3180

91050 ERLANGEN, GERMANY

Fax: +49 (0)9131 98 63 315 [documentation]

mailto:docu.motioncontrol@siemens.com

http://www.siemens.com/automation/service&support

Suggestions

Corrections

For Publication/Manual:

SINUMERIK 840D sl

HMI-Embedded

User documentation

From

Name:

Address of your company/department

Address: _____

Postal code: _____ City: _____

Phone: _____ / _____

Fax: _____ / _____

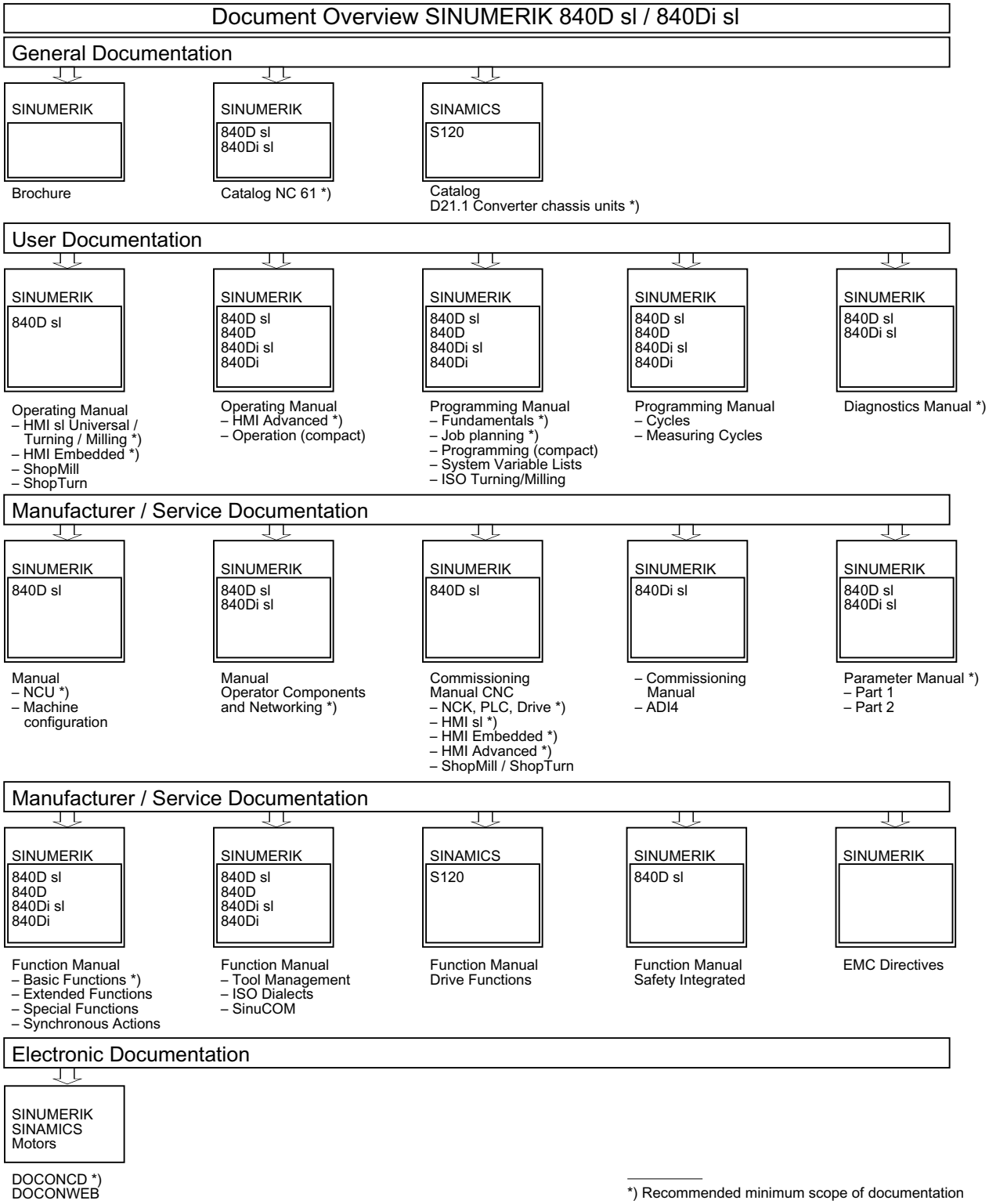
Operating Manual

Product code: 6FC5398-1AP10-2BA0
01/2008

Should you come across any printing errors when reading this publication, please notify us on this sheet. Suggestions for improvements are also welcomed.

Suggestions and/or corrections

Overview



*) Recommended minimum scope of documentation